CFturbo 10

User manual for CFturbo 10 software

CFturbo 10

Introduction

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

Table of Contents

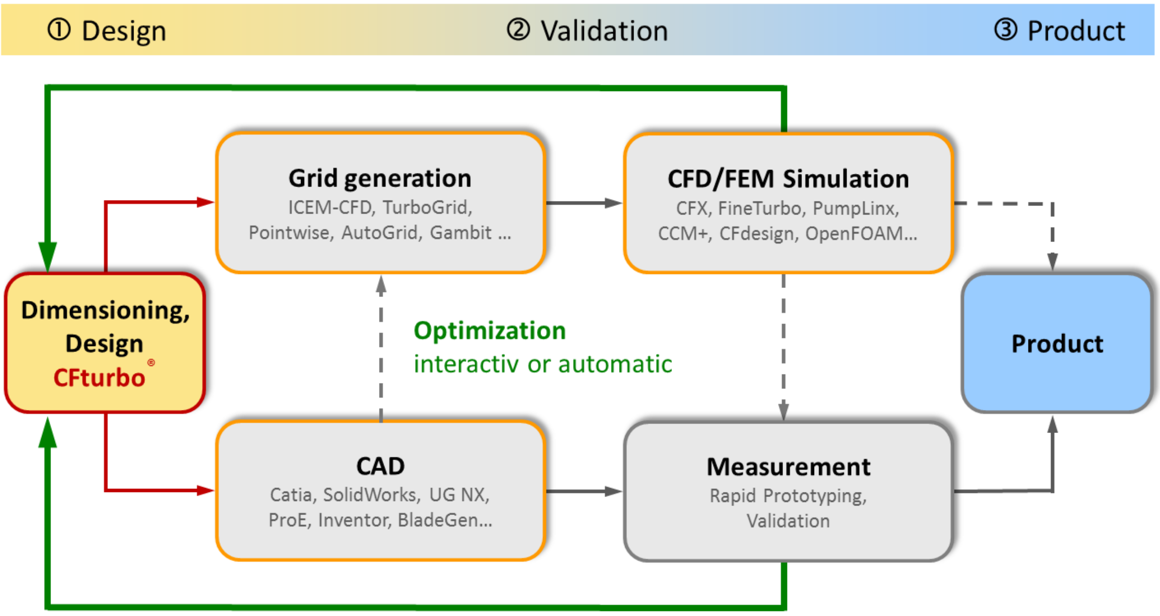
Part I

1 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.

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[454] before using the program.

Information about activating license you can read in chapter Licensing[12].

Contact persons you can find under Contact addresses 453, actual information on the CFturbo website.

Part II

2 General

This chapter contains some general program information about

2.1 Licensing

? Preferences | 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.

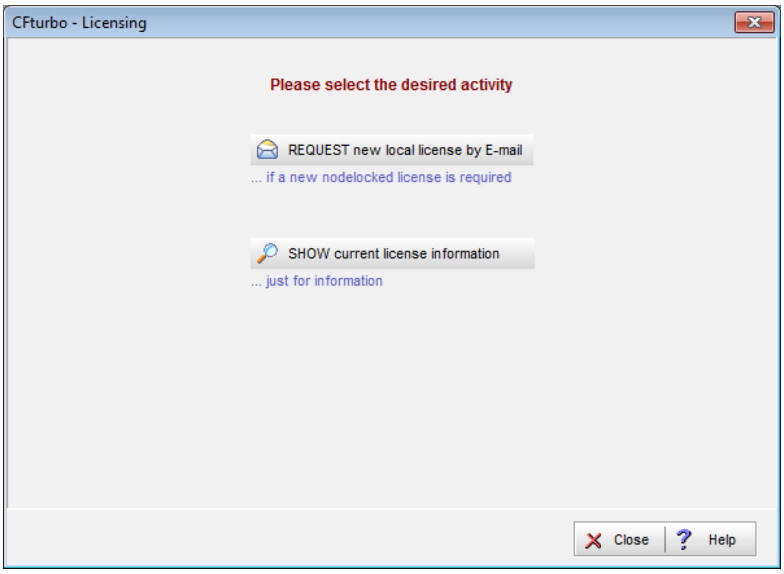
For example: A CFturbo project containing a stator, a radial pump impeller and a volute can always be opened. If only the modules for stator and radial pump impeller have been licensed, only this two components can be modified but not the volute.

A special feature of the CFturbo license model are stators. With every license for volute or radial impellers it is possible, to create and modify stators without blades.

		Module						
		Radial pump impeller	Radial ventilator impeller	Radial compressor impeller	Radial turbine impeller	Stator (vaned)	Stator	Volute
Present License:	Radial pump impeller	w	r	r	r	r	w	r
	Radial ventilator impeller	r	w	r	r	r	w	r
	Radial compressor impeller	r	r	w	r	r	w	r
	Radial turbine impeller	r	r	r	w	r	w	r
	Stator	r	r	r	r	w	w	r
	Volute	r	r	r	r	r	w	w

w = module data can be modified; r = module data is read only

Menu item **Licensing** enables license handling.



REQUEST 15 new license by e-mail

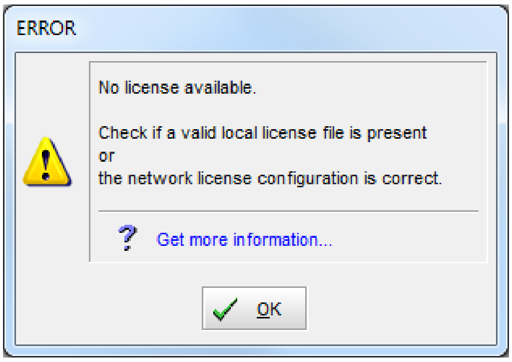
SHOW 23 current license information

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 Preferences | Settings | General 155.

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 local license has to be requested and installed

or

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

In general all licensing steps can be performed using remote desktop connection (RDP). But keep in mind that finally a Local Computer License can be used directly on this computer only and not via a RDP session. For this purpose, a Network Server License is required!

1. Local Computer License

Step	
1.	Start CFturbo - you see the "License" dialog 12 (or open menu Preferences | Licensing | Licensing).
2.	Request 15 local computer license and send license request to sales@cfturbo.com
3.	Save license file (<filename>.lic) received from CFturbo sales team to CFturbo installation directory (e.g. C:\Program Files (x86)\CFturbo 10)

4.	Show 23 license information to check modules and dates

2. Network Server License

(NOT available for trial license)

In advance of using CFturbo with a network license, the license server must be setup (includes requesting and installing a network license). For details see Network license setup 17.

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

Step	
1.	Configure computer for network license usage 17
2.	Start CFturbo and open menu Preferences | Licensing | Licensing
3.	Show 23 license information to check modules and dates

2.1.1 Local license setup

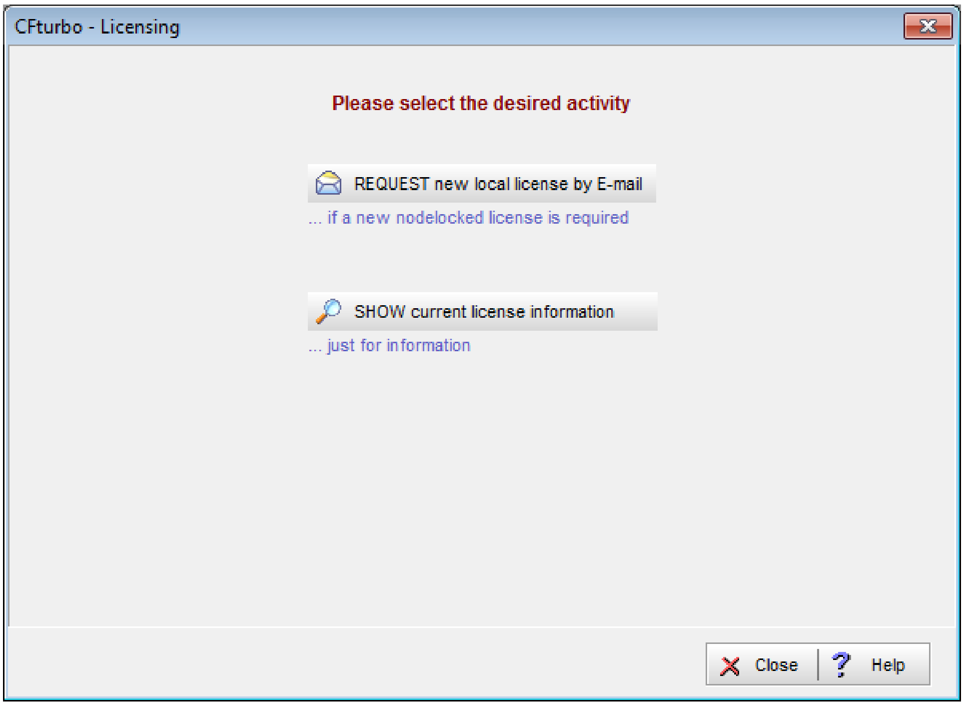
For using CFturbo with a local license 2 steps have to be performed:

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

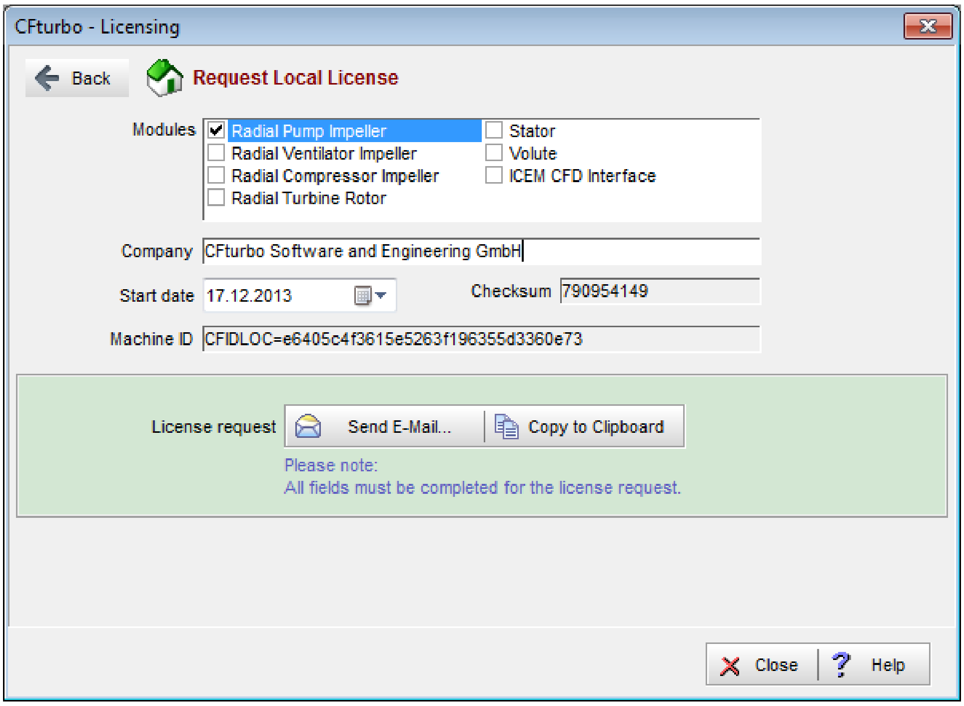
Note: If CFturbo is configured for using a network license 17, modules get checked out from that license first if available!

Requesting a local license

If not either a local license file is present or a network license is configured, CFturbo will start the licensing dialog (Preferences | Licensing | Licensing).



Here you can select **REQUEST new local license by E-mail**.



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

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

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

After input of all necessary information you can

- use the **Send 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 (e.g. C:\Program Files (x86)\CFturbo 10) 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 run CFturbo and check the license information 23.

2.1.2 Network 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 network 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 on a Virtual Machine environment is not tested and certified. 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 network license the following steps have to be performed:

1. Setting up the CFturbo license server 18
2. Requesting a license using the Request Generator 20
3. Storing the received license file in the CFturbo license server installation directory 18
4. Configuring the clients for accessing the network license 22

2.1.2.1 License server setup

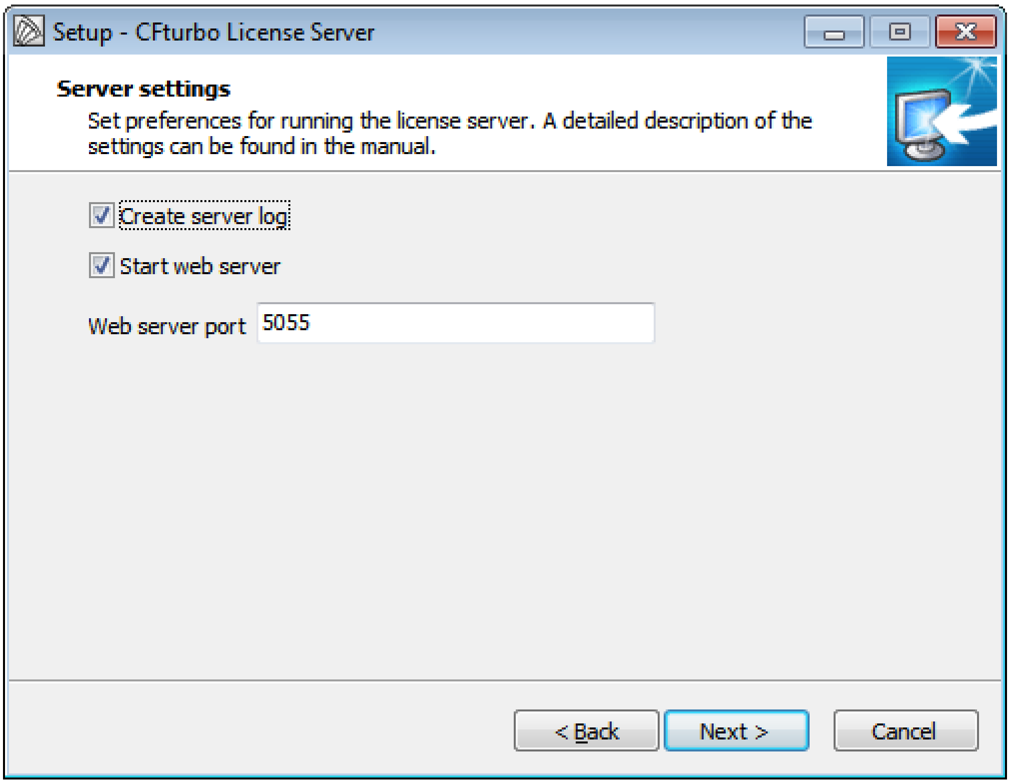
Installing the license server

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.

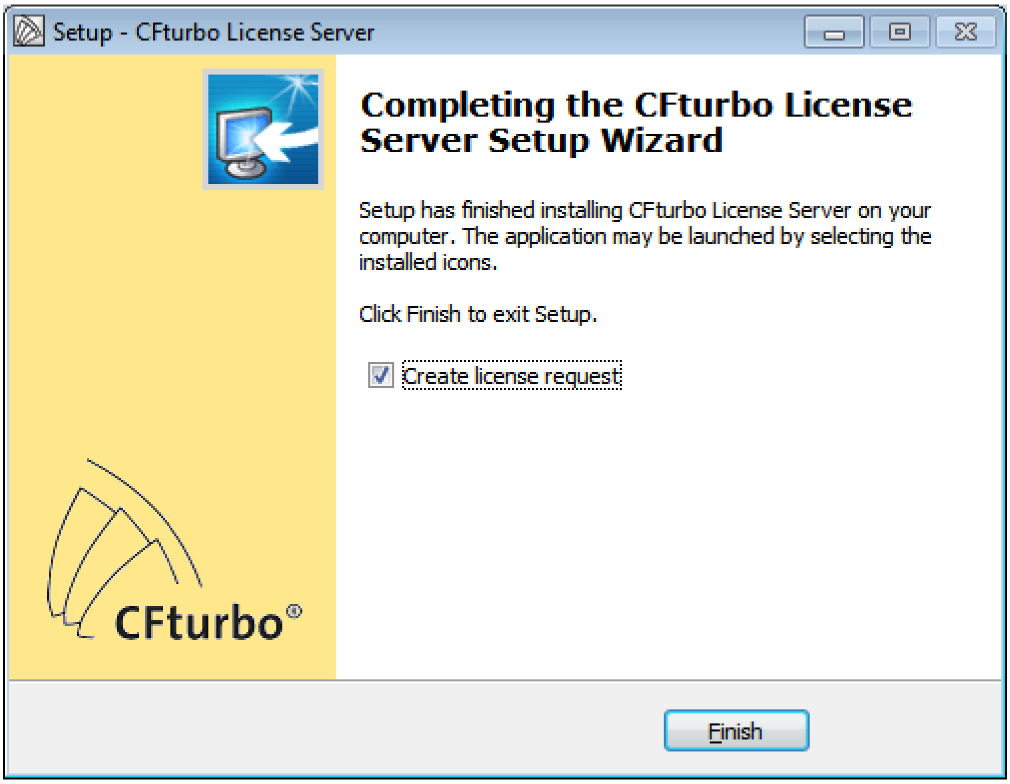
After running the setup and completing installation dir and start menu settings, the server parameters can be configured:



If **Create server log** is checked the server will write a logfile to the log directory. It is not recommended to disable this option!

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.

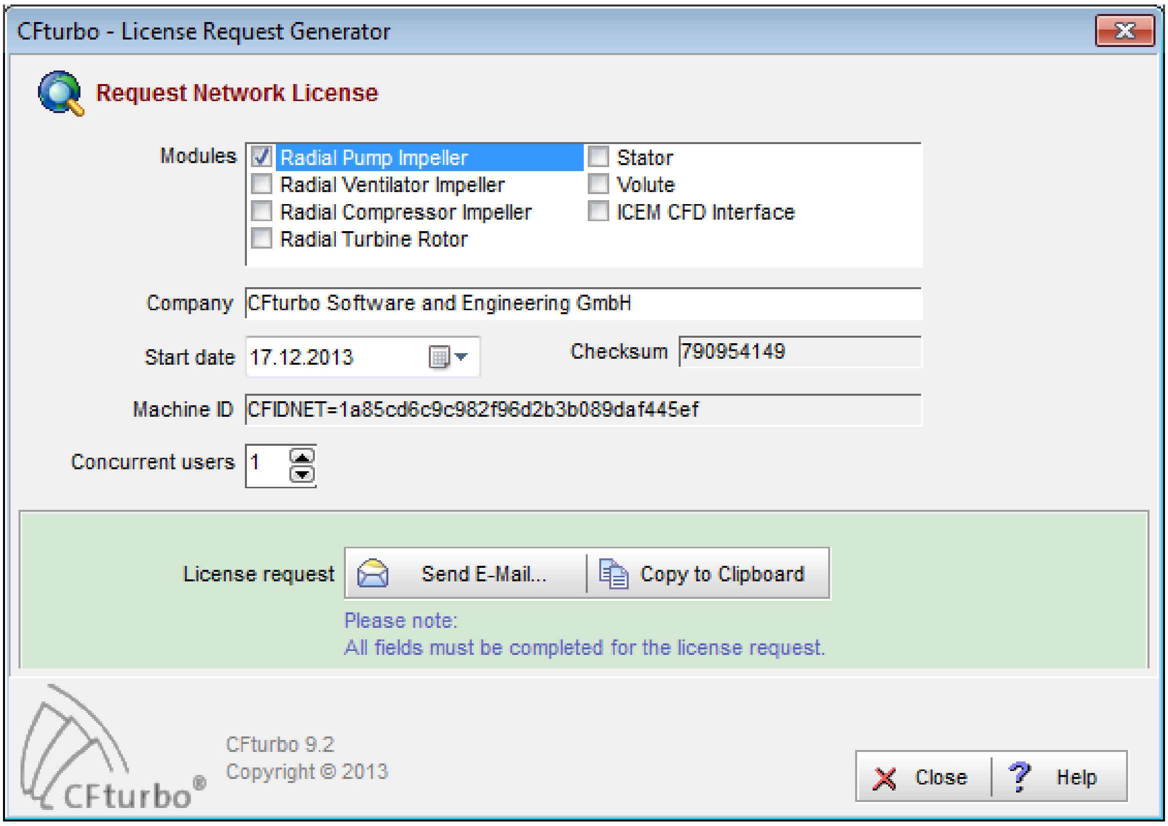
Note, that the setup is not checking for port conflicts, the port must be available. It can be changed e.g. by uninstalling and installing the server again.



The last wizard page offers to **Create a license request**. This option will start the Request Generator.

Requesting a network license

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



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

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

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

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

After input of all necessary information you can

- use the **Send 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"). Now the logfile and the web server page can be checked for the licenses to be running.

Firewall 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, if not configured in license file (on the SERVER or HOST line) defaults to port **5053**. The ISV server starts with a dynamic port number which is not known before startup 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 RepriseLM end user manual.

Additional configuration options

For additional configuration options check the RepriseLM end user manual.

2.1.2.2 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 RepriseLM end user manual.

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).

2.1.3 Show license information

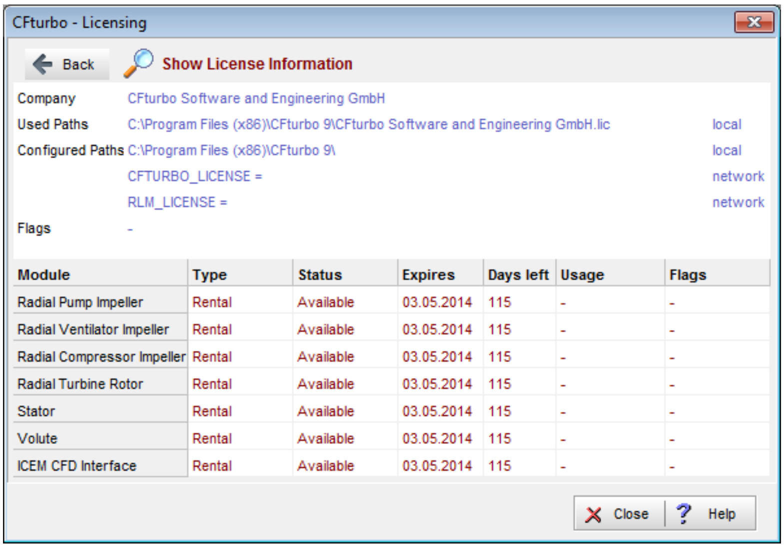
Current license information are displayed here.

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

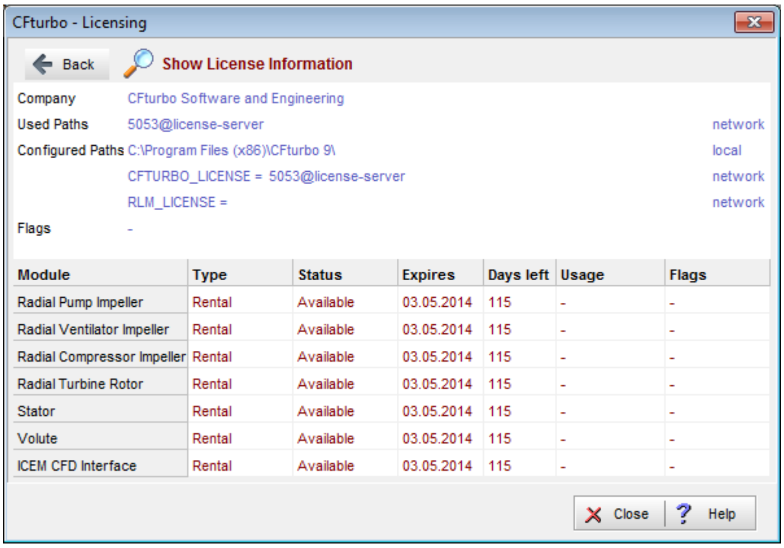
Path is the license file location and the content of the environment variables used for defining network license servers.

Normally **Flags** should not exist.

If available the last **Error** message of license checking is displayed.



Local license file is found and used

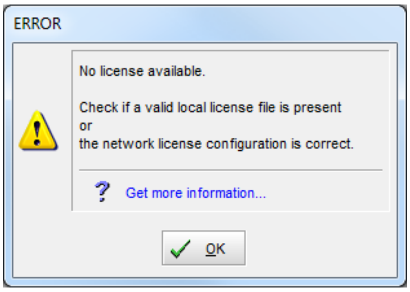


No local license file is found in program path, a network license path is configured

2.1.4 Troubleshooting

Error messages

Problem	Message	Reasons

No valid license available yet.		See Steps for licensing 12

Diagnostic configuration

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

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)

2.2 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>]

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 file (<batch file>).

Batch file

The **batch mode** of CFturbo is controlled by an XML file.
A template for a specific CFturbo project can be created via **Project | Export | Basic** 92 **| Batch mode template**.

Due to a close relation between the CFturbo file format and the batch mode format, only template/ batch mode files created with the same version as your CFturbo file should be used. After an update of CFturbo a new template can be exported and the needed adjustments can be done.

The resulting batch mode template contains all modifiable values of the CFturbo project as XML nodes supplemented by a short descriptions.

XML nodes of parameters that are not going to be changed can be deleted. The batch mode file also contains placeholder actions which must be completed with information related to file locations in the file system and export interface of the batch mode output.

File structure:

```
<?xml version="1.0" standalone="yes"?>
<CFturboFile Version="9">
     <CFturboBatchProject InputFile="<InputFileName>" />
           <Updates>
                 [...]
           </Updates>
           <BatchAction .../>
     </CFturboBatchProject>
</CFturboFile>
```

A batch-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 templates 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 absolute path of the CFturbo project file.

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** 85.

Attribute	Value	optional	Description

Name	Export	no	Name of action
ExportInterface	e.g. "General"	no	Export interface to use The following values are valid: AutoCAD, BatchTemplate, BREP, Catia, General, GeneralXML, IGES, Inventor, NumecaAG, NumecaIGG, PerformanceData, Pointwise, PumpLinx, Report, SolidWorks, StarCCM, STL, TetraVolMesh, 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. The list is created when the batch mode template for the project is created and should be modified on this base.

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

  Is used for saving the CFturbo project after applying batch updates.
  Can also be used for the automatic conversion of CFturbo files created with older program versions.
  The OutputFile attribute specifies the absolute path of the file save destination.

For details about component-specific parameters see:

→ Parameters for impellers 29

→ Parameters for volutes 33

If certain values are not in the batch mode template that are listed there as available on the sub pages, it may be due to them being meaningless in the context of the current project settings. In this case they are not included in the batch mode template. (For example values related to splitter blades, if splitter blades are not enabled).

2.2.1 Parameters for impellers/ stators

XML Tag (+attributes)			Description	Unit
Main dimensions <MainDimensionsElement>				
Impeller		<dN>	Hub diameter d_H at inlet	m
		<dS> [1]	Suction diameter d_S at inlet	m
	axial impellers	<dH2> [1]	Hub diameter d_{H2} at outlet	m
		<dS2> [1]	Shroud diameter d_{S2} at outlet	m
	radial/ mixed-flow impellers	<d1> [1]	Inlet diameter (leading edge) d_1	m
		<b1> [1]	Inlet width (leading edge) b_1	m
		<d2> [1]	Impeller diameter d_2	m
		<b2> [1]	Impeller outlet width b_2	m
		<xTip>	tip clearance (for unshrouded impellers)	m
Stator		<MerData> ("MerInlet", "MerOutlet")	Inlet/Outlet geometry (see Interface definition 40): - Interface position Hub/ Shroud if the inlet/outlet is the primary interface side - Offsets for Hub/ Shroud or Center line. Used to define the absolute geometrical position.	m
Meridional contour <Meridian>				

XML Tag (+attributes)	Description	Unit
<Bezier4MerLE Name="GeoLeadingEdge"> <u-Hub> <u-Shroud>	Leading edge position on hub (0…1) Leading edge position on shroud (0…1) These value take higher priority than control points of the edge curve below and hence override the first and last control point values	-
<Bezier4MerLE Name="GeoLeadingEdge"> <Points>	Control points of leading edge curve. Number of control points depends on selected curve mode.[4] see meridional contour > leading/ trailing edge 284	-
<Bezier4MerTE> <u-Hub> <u-Shroud>	Trailing edge position on hub (0…1) Trailing edge position on shroud (0…1) These value take higher priority than control points of the edge curve below and hence override the first and last control point values Only available if trailing edge is not fixed to outlet	-
<Bezier4MerTE> <Points>	Control points of trailing edge curve. Number of control points depends on selected curve mode.[4] see meridional contour > leading/ trailing edge 284 Only available if trailing edge is not fixed to outlet	-
<Bezier4MerLE Name="GeoSplitLeadingEdge"> <u-Shroud> <u-Hub>	Splitter leading edge position on hub (0…1) Splitter leading edge position on shroud (0…1) These value take higher priority than control points of the edge curve below and hence override the first and last control point values	-

XML Tag (+attributes)	Description	Unit
<Bezier4MerLE Name="GeoSplitLeadingEdge"> <Points>	Control points of splitter leading edge curve. Number of control points depends on selected curve mode.[4] see meridional contour > leading/ trailing edge 284	-
<ListObjectBezier4Mer Name="GeoHub">	Contour segment of Hub contour containing a set of control points. The number of control points depends on the selected curve mode.[4] see meridional contour > Hub-Shroud contour 274 (only available for Hub-Shroud design mode)	-
<ListObjectBezier4Mer Name="GeoShroud">	Contour segment of Shroud contour containing a set of control points. The number of control points depends on the selected curve mode.[4] see meridional contour > Hub-Shroud contour 274 (only available for Hub-Shroud design mode)	-
<ListObjectBezier4Mer Name="GeoMiddleLine">	Midline contour containing a set of control points. The number of control points depends on the selected curve mode.[4] see meridional contour > Design Modes 269 (only available for Midline design mode)	-
Blade properties <BladeProperties>		
<nBl>	Number of blades n_{Bl}	-
<Count>	Number of blade profiles	-
<Beta1 Blade="0"> [2]	Blade angles at leading edge β_1 for each blade profile	rad

XML Tag (+attributes)	Description	Unit
<Beta2 Blade="0"> [2]	Blade angles at trailing edge β_2 for each blade profile	rad
<Beta1 Blade="1"> [2 3]	Splitter blade angles at leading edge $\beta_{1,Spl}$ for each blade profile	rad
<Beta2 Blade="1"> [2 3]	Splitter blade angles at trailing edge $\beta_{2,Spl}$ for each blade profile	rad
<s1 Blade="0">	Main blade thickness - on small radius (LE) [Hub,Shroud]	m
<s2 Blade="0">	Main blade thickness - on large radius (TE) [Hub,Shroud]	m
<s1 Blade="1">	Splitter blade thickness - on small radius (LE) [Hub,Shroud]	m
<s2 Blade="1">	Splitter blade thickness - on large radius (LE) [Hub,Shroud]	m
<inc_RQ>	Incidence - flow ratio Q_shockless/ Q_BEP [Hub,Shroud]	%
<inc_i>	Incidence angle [Hub,Shroud]	rad
Mean lines <SkeletonLines>		
<RelativeSplitterPosition>	Splitter trailing edge position (tangential) between neighboring main blades	%
<Bezier3SL> [3 4]	m,t-Bezier control points to modify wrap angle and blade shape	-
<BezierBetaSL> [3 4]	Bezier points of β distribution for indirect modification of blade shape	-
Blade profiles <BladeProfiles>		
<BezFillProf Name="MBl">	Blade thickness distribution along **main** blade profiles. Bezier curves for pressure- & suction	-

XML Tag (+attributes)	Description	Unit
	side [PS, SS])	
<BezFillProf Name="SBl">	Blade thickness distribution along **splitter** blade profiles. Bezier curves for pressure- & suction side [PS, SS])	-

[1] Make sure that main dimensions are not calculated automatically (see impeller main dimensions 201) to make these values available in batch mode. Save these changes into the Project file before applying batch mode updates.

[2] Make sure that 'automatic blade angle update' is deactivated in the blade property dialog 307 to make blade angles available in batch mode. Save these changes into the Project file before applying batch mode updates.

[3] Values for splitter blades are only available when splitters are not geometrically linked to main blades. See blade properties 296.

[4] Control points are always listed as Cartesian coordinates. They can be modified within the same constraints that exist in interactive design mode (Modifications that violate the constraints will be corrected).
Explicit coordinates will also be overridden when additional relative coordinates for corresponding control points are provided. These relative parameters are listed below the complete control point list and are specified the same way as in the corresponding design dialogs (see Right Click on Beziér control point 44).

2.2.2 Parameters for volutes

XML Tag (+attributes)	Description	Unit
Inlet definition <SpiralCasingBC>		
<FQ>	Flow factor F_Q	-
<MerData>	Inlet geometry (see Interface definition 40): - Interface position Hub/ Shroud if the volute is the primary interface side - Offsets for Hub/ Shroud or Center line. Used to change Inlet diameter (d_4) and Inlet width (b_4).	m
Diffuser <SpiralCasingDiff>		

XML Tag (+attributes)		Description	Unit
<Bezier4Diff> <H6>		Diffuser height (h6). (See Diffuser 428)	m
<Diameter> or <Rectangle>		Dimensions of the 'End cross-section'. Depending on the used shape it either specifies a Diameter for circular end cross-sections or width and height for rectangular end cross sections (See Diffuser 428)	m
Cut-water <SpiralCasingCutwater>			
Simple	<PhiT0>	Angular position $\varphi_{C,0}$ (see Simple Cut-water 437)	rad
Fillet	<Fillet>	Fillet radius R (see Fillet Cut-water 440)	m
	<DiffBase FormFactor>	Diffuser Base Form factor (see Fillet Cut-water 440)	-
	<PhiT0>	Spiral start position (see Fillet Cut-water 440)	rad
Double volute	<EllLERatio>	Splitter edge ratio (see Cut-water 434)	-

2.2.3 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.
1	Last batch run was completed with warnings but no errors.
2	Last batch run was completed with errors.

2.2.4 Example

The example of a CFturbo batch file below, changes the blade number of the Pump1 example project.

Subsequently the modified project gets exported as geometry export as well as saved into the CFturbo project file "Pump1_mod.cft".

```
<?xml version="1.0" standalone="yes"?>
<CFturboFile Version="9">
  <CFturboBatchProject InputFile="C:\Testing\Pump1.cft">
    <Updates>
      <CFturboProject Type="Object">
        <CFturboDesign_RadialImpeller Type="Object" Name="&lt;Radial
Impeller&gt;" Info="Cfturbo GmbH" Index="0" Desc="CFturbo component">
          <BladeProperties Type="Object" Desc="Blade properties">
            <nBl Type="Integer" Desc="Number of blades">7</nBl>
          </BladeProperties>
        </CFturboDesign_RadialImpeller>
      </CFturboProject>
    </Updates>
    <BatchAction Name="Export" ExportInterface="General" WorkingDir="C:\Testing
\" BaseFileName="Pump1_9.1_all" AllComponents="1"/>
    <BatchAction Name="Export" ExportInterface="General" WorkingDir="C:\Testing
\" BaseFileName="Pump1_9.1">
      <ExportComponents>
        <Value Type="Integer">1</Value>
      </ExportComponents>
    </BatchAction>
    <BatchAction Name="Export" ExportInterface="STEP" BaseFileName="Pump1_9.1"
ModelState="Solids only">
    </BatchAction>
    <BatchAction Name="Save" OutputFile="pump1_mod.cft"/>
  </CFturboBatchProject>
</CFturboFile>
```

During runtime a log-file <batch file>.log is created in the directory of <batch file>:

```
29.10.2013 16:29:42 [INFO]       CFturbo 9.2  -  29.10.2013
29.10.2013 16:29:42 [INFO]       Time:              29.10.2013
16:29:42
29.10.2013 16:29:42 [INFO]       File:              c:\Testing
\pump1_m.cft-batch
29.10.2013 16:29:42 [INFO]       Logfile:           c:\Testing
\pump1_m.log
29.10.2013 16:29:42 [INFO]       Working directory: C:\Program Files
(x86)\CFturbo 9
29.10.2013 16:29:42 [INFO]       ***
29.10.2013 16:29:42 [INFO]       Reading batch file: c:\Testing
\pump1.cft-batch
29.10.2013 16:29:42 [INFO]       Starting batchproject for input
file: C:\Testing\Pump1.cft
29.10.2013 16:29:42 [INFO]       Open input file: C:\Testing
\Pump1.cft
29.10.2013 16:29:42 [INFO]       Update design parameters
29.10.2013 16:29:42 [INFO]       Running geometry update with data:
29.10.2013 16:29:42 [INFO]       <CFturboProject Type="Object">
     <CFturboDesign_RadialImpeller Type="Object" Name="&lt;Radial
Impeller&gt;" Info="CFturbo Software and Engineering GmbH - cft-senb1
(2/4/24)" Index="0" Desc="CFturbo component">
<BladeProperties Type="Object" Desc="Blade properties">
     <nBl Type="Integer" Desc="Number of blades">7</nBl>         </
BladeProperties>       </CFturboDesign_RadialImpeller>  </
CFturboProject>
29.10.2013 16:29:42 [INFO]       Run design steps
29.10.2013 16:29:43 [INFO]       No hints.
29.10.2013 16:29:43 [INFO]        1: <Radial Impeller>: Blade
properties: Blade angles are updated automatically. Therefore
geometry modifications are possible.
29.10.2013 16:29:43 [INFO]        1: <Radial Impeller>: Model
finishing: currently NOT up-to-date
```

```
29.10.2013 16:29:43 [INFO]       Export-action found for format:
General
29.10.2013 16:29:43 [INFO]       Selecting all (1) components for
export!
29.10.2013 16:29:43 [INFO]       Saving export files successful,
export log:
29.10.2013 16:29:43 [INFO]       29.10.2013 16:29:43 [INFO]
File: C:\Testing\Pump1_9.1_all.cft-geo successfully exported
29.10.2013 16:29:43 [INFO]       Export-action found for format:
General
29.10.2013 16:29:43 [INFO]       Saving export files successful,
export log:
29.10.2013 16:29:43 [INFO]       29.10.2013 16:29:43 [INFO]
File: C:\Testing\Pump1_9.1.cft-geo successfully exported
29.10.2013 16:29:43 [INFO]       Export-action found for format: STEP
29.10.2013 16:29:43 [INFO]       No working directory set, using
default: C:\Testing\
29.10.2013 16:29:45 [INFO]       Run trimming
29.10.2013 16:29:47 [INFO]       Run fillet creation
29.10.2013 16:30:48 [INFO]       Saving export files successful,
export log:
29.10.2013 16:30:48 [INFO]       29.10.2013 16:30:43 [INFO]
Updated 3D data
29.10.2013 16:30:48 [INFO]       29.10.2013 16:30:43 [INFO]
Setting model state: Solids only
29.10.2013 16:30:48 [INFO]       29.10.2013 16:30:48 [INFO]
File: C:\Testing\Pump1_9.1.stp successfully exported
29.10.2013 16:30:48 [INFO]       Save output file: c:\Testing
\pump1_mod.cft
29.10.2013 16:30:48 [INFO]       ***
29.10.2013 16:30:48 [INFO]       Batch mode terminated. (01:08.160
min)
```

2.3 Project structure and interfaces

A CFturbo project describes a complete single-stage machine or a single stage of a multi-stage machine. Flow-conducting parts of the machine can be designed by CFturbo.

Project types

The follwing project/ machine types are available:

- Pump
- Ventilator
- Compressor
- Turbine

Project structure

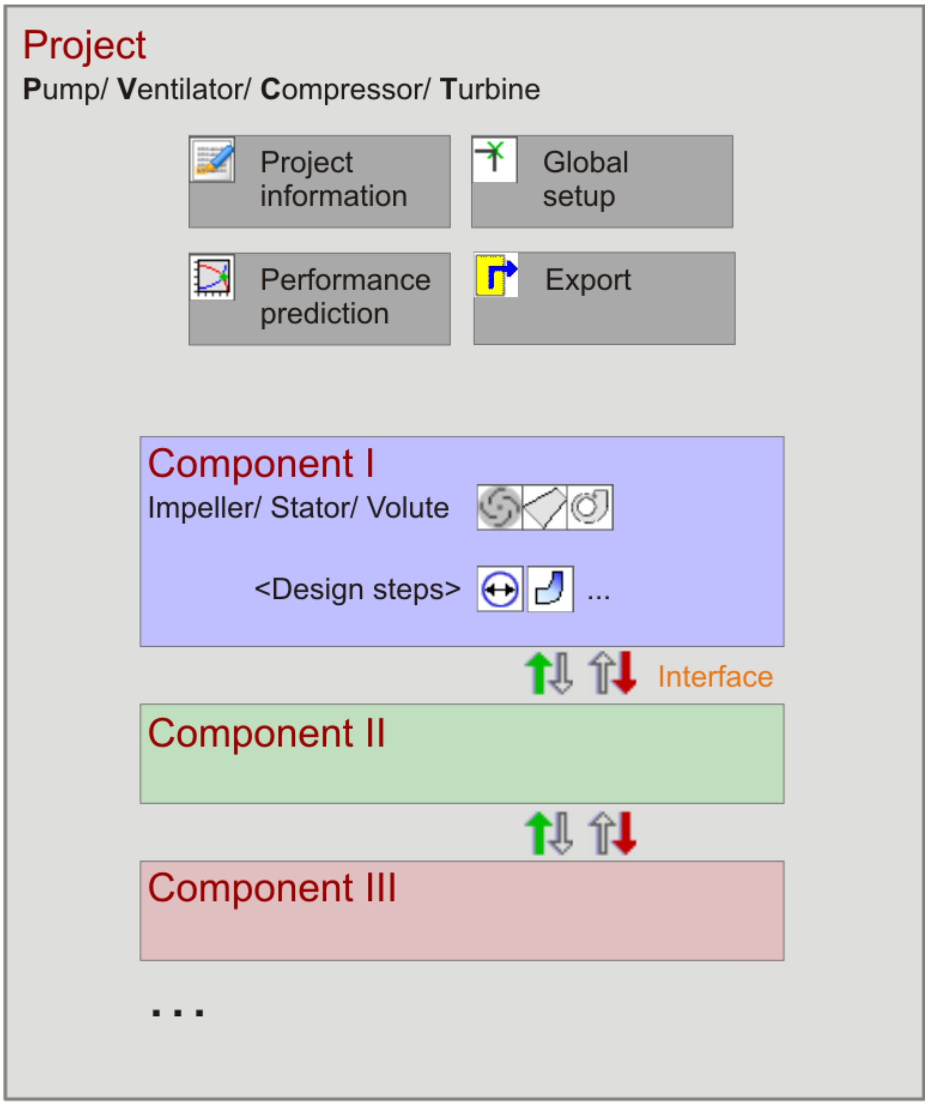
A project consists of the global parts

- Project information 71
- Global setup 71
- Performance prediction 77
- Exp 85 ort 85

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

- 1 or 2 Impellers on any position
- 1 Volute as last component
- any number of Stators (vaned or unvaned)

Components can be added directly in the components view 169 or via the project menu 140.



Interfaces between components

Interfaces exist between neighboring components describing their coupling. 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**

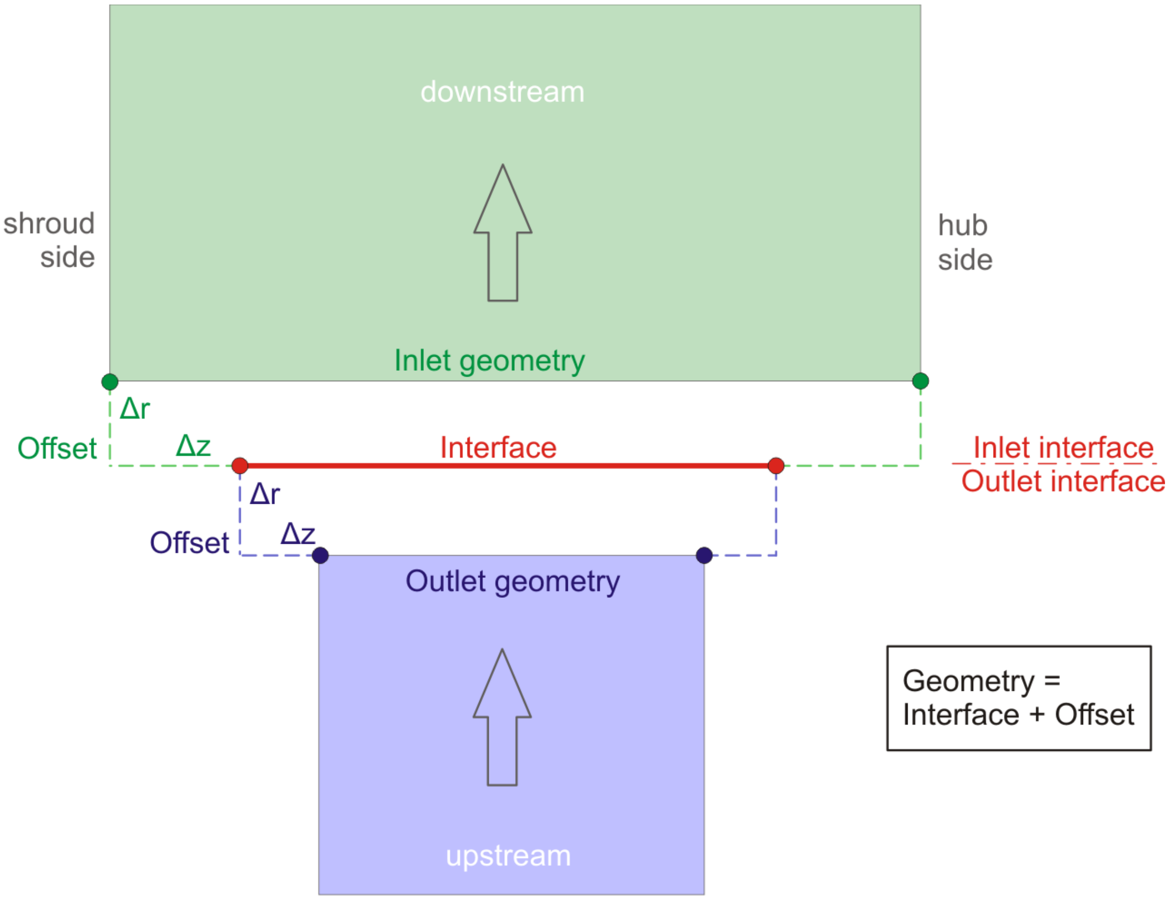
	Outlet cross section of a component is defined by the inlet cross section of next component.

Interface coupling can be adjusted in the component view 168 directly at the interface position between neighboring components.

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

2.3.1 Interface definition

The sketch illustrates the general layout of an interface between 2 neighboring components:



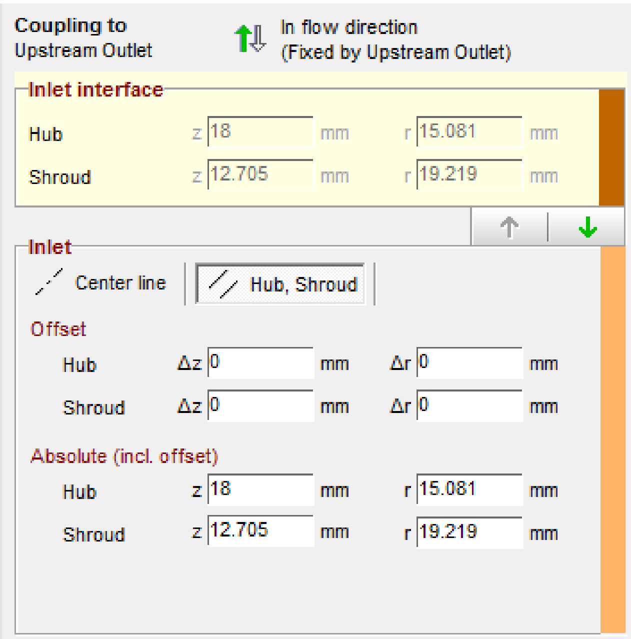
Primary / Secondary

One side (component) of the interface is primary always, the other one is secondary. The primary side determines the position of the interface (red in the sketch), the secondary has to align on the primary side. Each interface side can define an offset to the interface optionally.

If the geometry of the primary component and therefore the position of the interface is changing, then the component with the secondary interface is adjusted automatically. If a component is deactivated (see Active/ Rename/ Delete 141), then no adjustment will be effected - therefore an overlapping of neighboring components is possible, which is illustrated by a warning (see Components 168).

Interface definition

The interface definition at volute inlet 401 as well as at stator inlet 389 and outlet 390 is made in an uniform manner.



Coupling
Information to interface coupling direction

Inlet/ outlet interface
Interfaces position at hub and shroud side (deactivated for secondary interface side)

↑ ↓
Coordinate transfer from geometry to interface and reverse

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 368 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 interface 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 71.

Possible warnings

Problem	Possible solutions
Invalid inlet/ outlet interface.	
Intersection between interface and geometry detected.	Check interface definition of the component.
No matching inlet/ outlet interface (considering outlet extension [of previous impeller])	
2 neighboring components are not matching on their interface.	Check both sides (components) of the interface if the hub and shroud points are identical. On the inlet interface: if the previous component is an impeller then the outlet extension 368 of this impeller can cause the problem. On the outlet interface: if this component is an impeller then the outlet extension 368 can cause the problem.

2.3.2 Automatic calculations

Some component design steps contain automatic calculations. 

Currently these are:

- Impeller main dimensions 190: calculation of dimension values
- Impeller blade properties: calculation of blade angles 307 β_{B1}, β_{B1} (meanline design mode) or Profile properties 359 (airfoil/ hydrofoil design mode)

These automatic calculations can be activated or deactivated. Both approaches have their specific advantages and disadvantages:

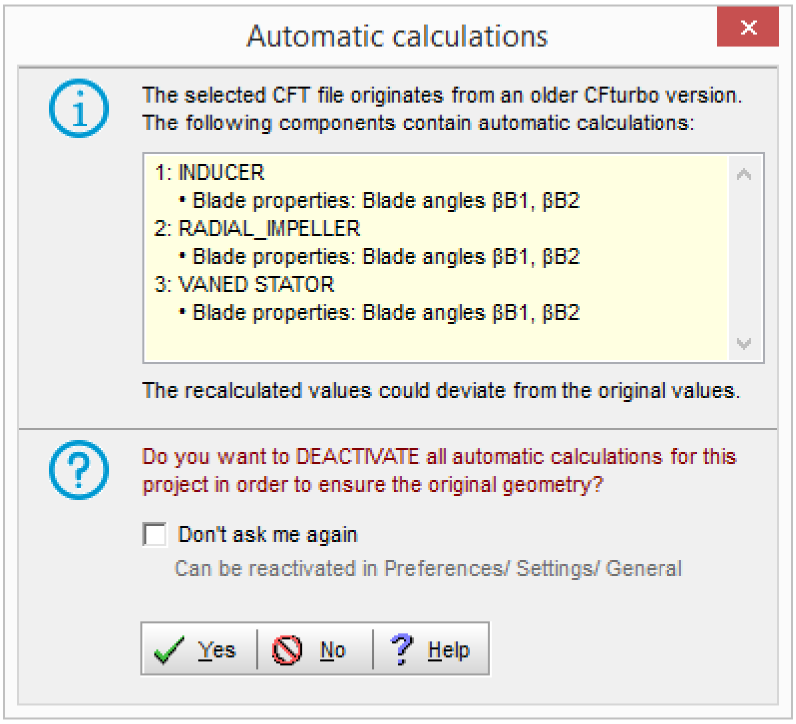
- **Automatic calculation**:
  It's assured that the calculation results are up-to-date based on the latest input parameters.

The formerly used values could be 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.

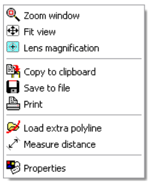


2.4 Graphical dialogs

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:

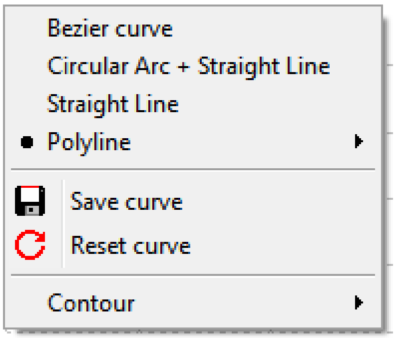


- Zoom window by mouse
- Fit view
- Lens magnification
- Copy to clipboard
- Save diagram as BMP, GIF, JPG, PNG or WMF
- Print
- Add any polyline from file (x,y points) to compare different curves
- Measure distance
- Configure diagram

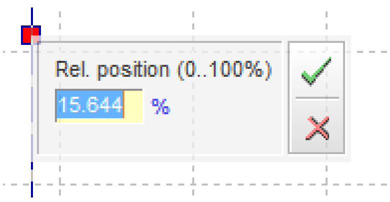
Context sensitive popup menus

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.



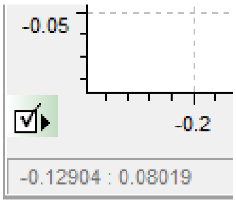
For Bezier curves popup menus are available for special actions concerning the curve.

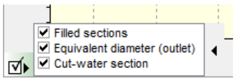


An alternate method to specifying Bezier points by the mouse, you may enter the accurate coordinates of Bezier points in a small dialog window that appears by clicking the right mouse button on the chosen Bezier point.

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.

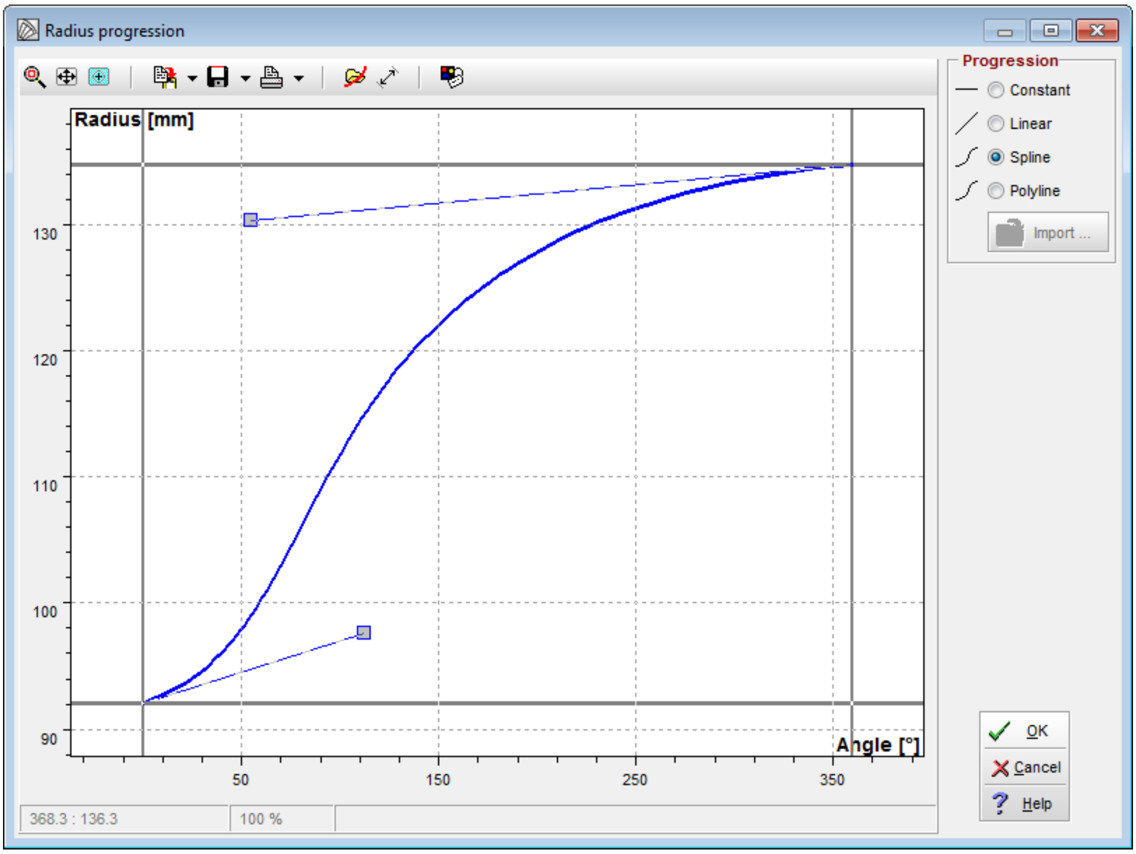
The state of each display option is saved internally and restored next time.

Miscellaneous

- Coordinates of mouse cursor are displayed in format x:y bottom left in the status bar.
- Position and size of dialogs are saved to restore it in the same way when they are called again.
- If CFturbo generates primary design automatically you may see Initial design on the top right of the diagram.
- If numerical values are entered in tables, then a new value is only activated and the diagram is updated if the <Enter> key is pressed or a new cell of the table is selected.

2.5 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 268
- Angular positions, in Blade mean lines 319
- Spiral cross section progression, in Spiral development areas 417

Import Polyline

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

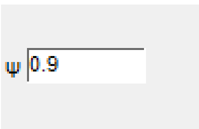
Text file format:

`# cross section distribution`
`# start/end tangential,`
`# midsection linear`
`# (spline interpolation 9 points)`
`0.00 0.00000`
`0.04 0.01728`
`0.08 0.03830`
`0.12 0.06368`
`0.16 0.09404`
`0.20 0.13000`
`0.24 0.17164`
`0.28 0.21687`
`0.32 0.26314`
`0.36 0.31018`
`0.40 0.36000`
`0.44 0.41404`
`0.48 0.47102`
`0.52 0.52898`
`0.56 0.58596`
`0.60 0.64000`
`0.64 0.68982`
`0.68 0.73686`
`0.72 0.78313`
`0.76 0.82836`
`0.80 0.87000`
`0.84 0.90596`
`0.88 0.93632`
`0.92 0.96170`
`0.96 0.98272`
`1.00 1.00000`
	• All lines starting with a "#" symbol are comments. All other lines contain the numerical values.
	
	• x and y coordinate values can be separated by "*comma*", "*semicolon*", "*space*" or "*tabulator*".
	
	• "*Dot*" character is required to be used as decimal separator.
	
	• Values are imported in the currently active units of the diagram axes.
	
	• The file can have any or no filename extension.
	
	A sample file can be generated by right clicking the progression curve and selecting "Save polyline".

2.6 Edit fields with empirical functions

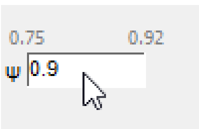
Some edit fields are connected with empirical functions 145. This becomes visible when activating the edit field by mouse click.

Default



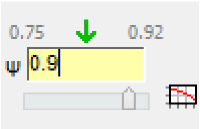
Default appearance of edit field.

Mouse-over



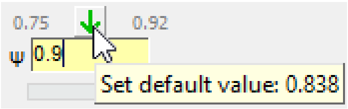
Appearance if the mouse cursor is over the edit field. Min. and max. values are displayed if a recommended range exists.

Focused



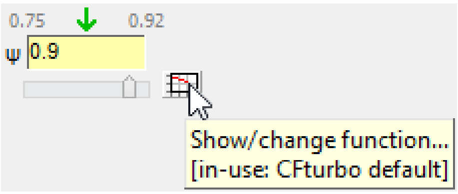
Appearance, if the edit field is focused (mouse click into the field). If a recommended range exists, min. and max. values are displayed as well as a sliding bar below.

Default value



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

Empirical function



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

2.7 Troubleshooting

This chapter provides information on how problems can be handled:

2.7.1 Error reporting

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

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

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

- Send bug report

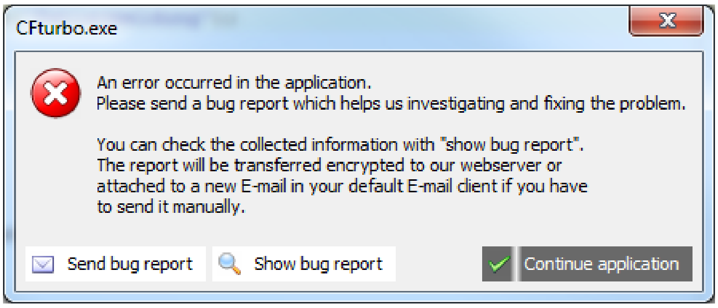
  Follow the *Send assistant* to add user and contact information as well as configuring the bug report. Finally, the report will be sent to our web server encrypted.

- Show bug report

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

- Continue application (Default)

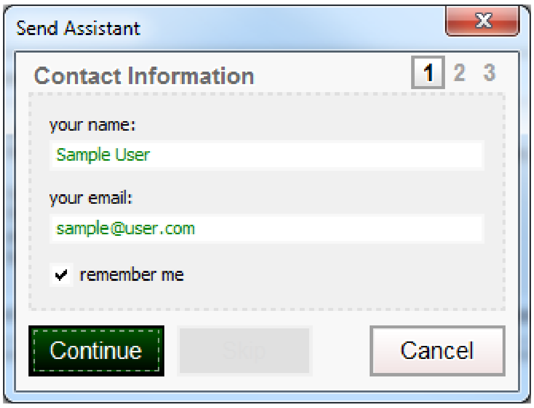
Continue working with CFturbo without sending the bug report.



Send assistant

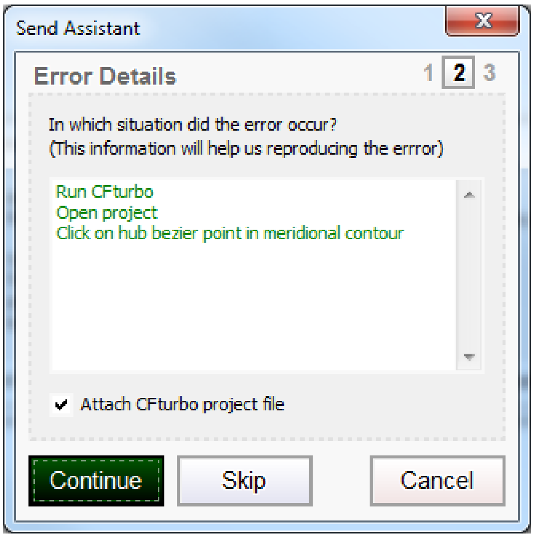
The Send assistant will guide you sending the bug report.

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

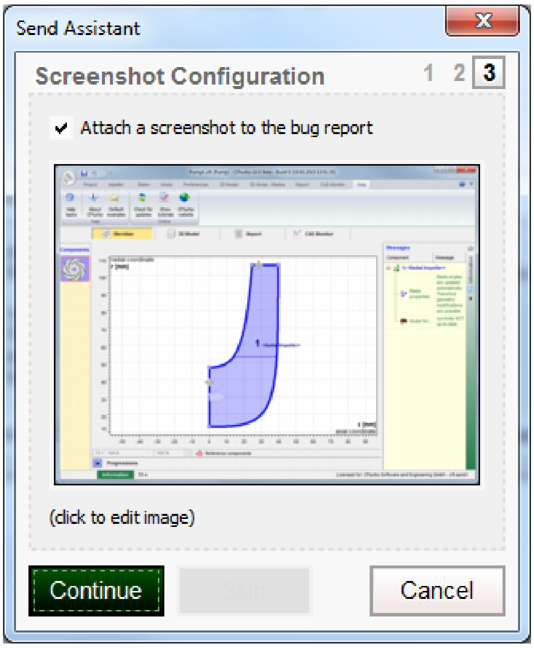


The second step asks you for the details of the situation, the error occurred in. Please note that it is extremely helpful if the error can be reproduced.

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



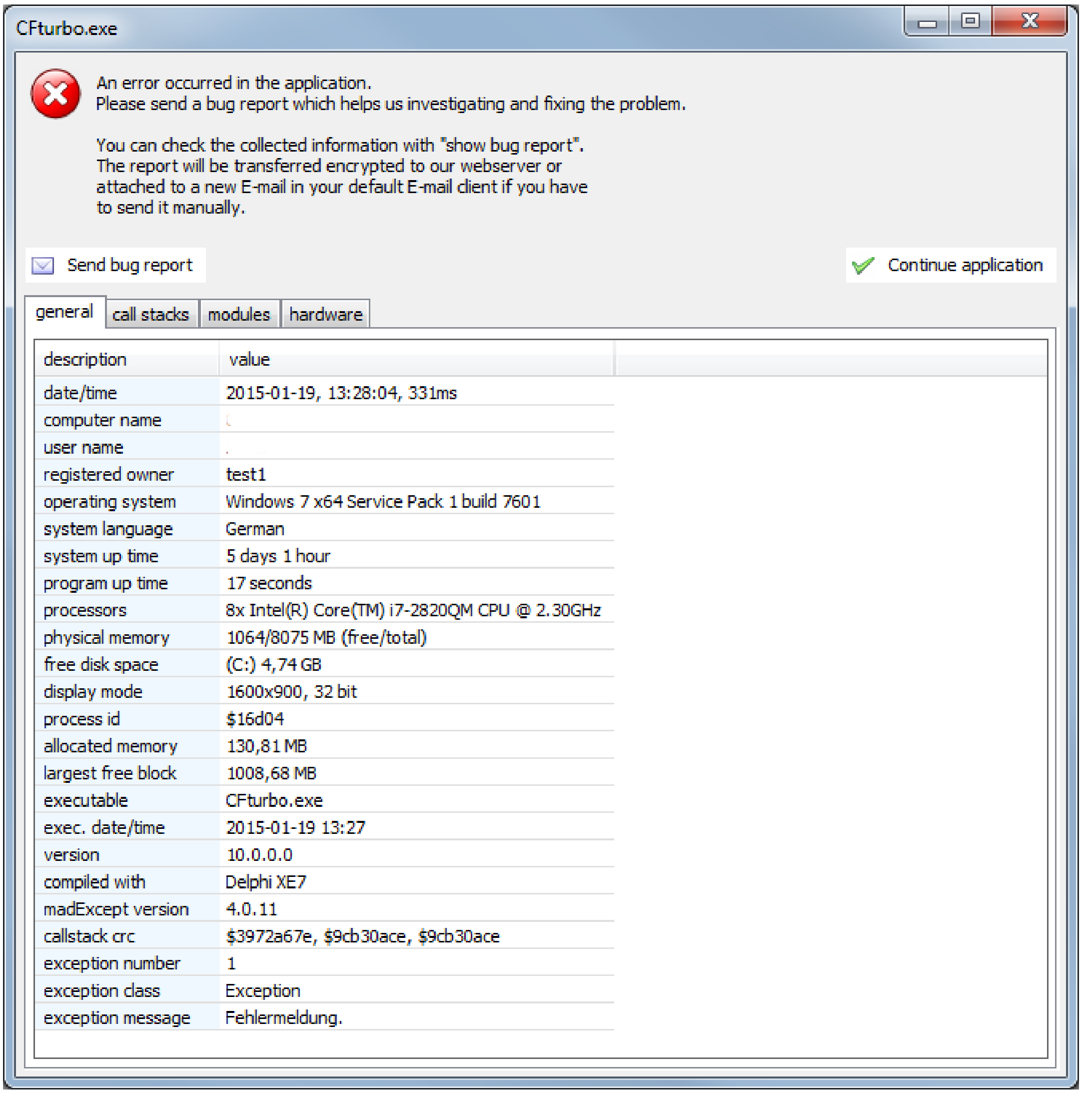
Finally you can choose if a screenshot should be attached. If Continue is clicked, the report will be sent encrypted to our web server.



If automatic sending fails, e.g. due to missing network connection, a mail with all details and attachments will open in your default mail client and you have to send it manually.

Detail view

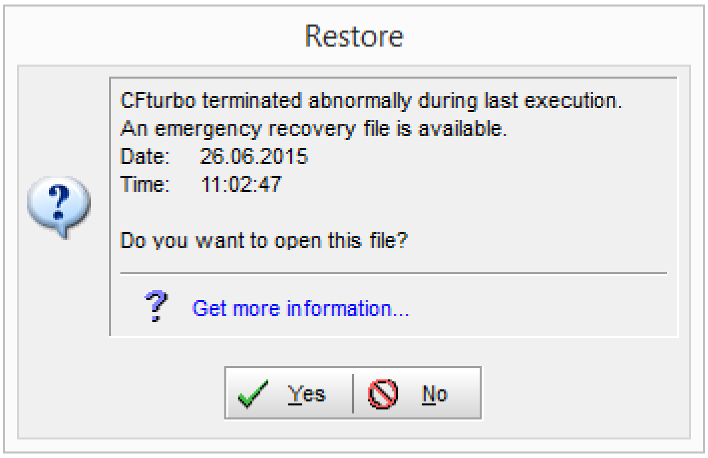
The detail view shows you the information that is collected about the error and the current state of CFturbo. Also basic system information is included.



2.7.2 Emergency recovery

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

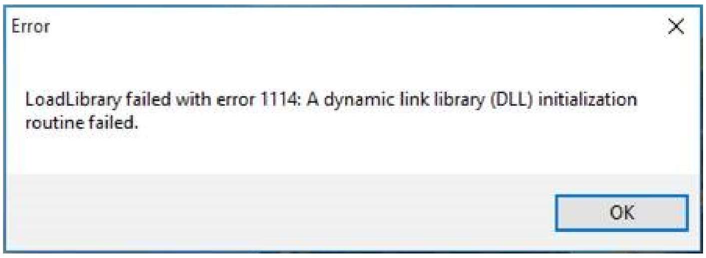
In this case the following message is displayed and one can open this last project state optionally.



The last project state is the newest item of the Undo 139 list of the previous project.

2.7.3 Known problems

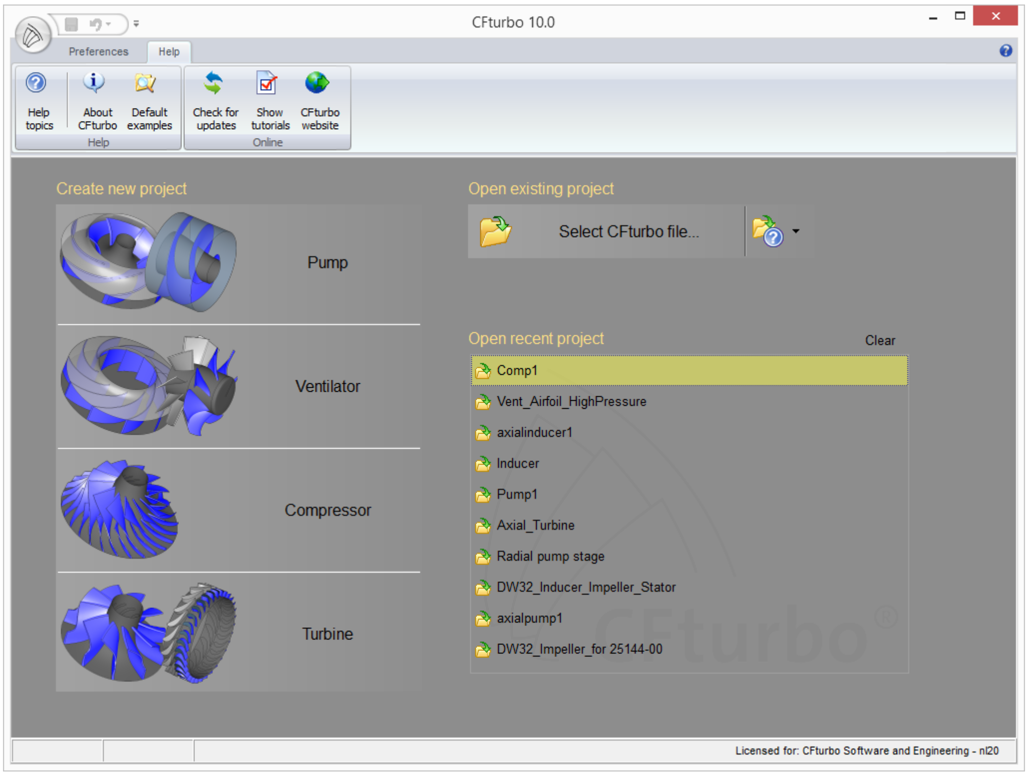
The following table lists known problems together with their possible solutions:

Problem	Possible solutions
When CFturbo is started, the following error message is displayed:  **LoadLibrary failed on Windows 10**	Update the graphics card driver.

Part III

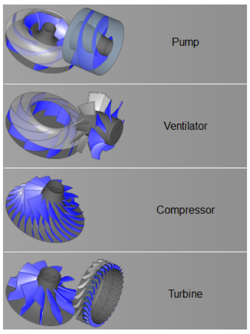
3 Start

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 67.

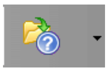
After creating a new project the Global Setup 71 dialog is starting automatically.

Afterwards several components can be added 140 to the project.

Open existing project

Here you can select existing projects:

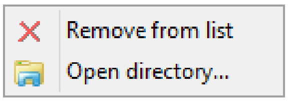
Open any CFturbo project (*.cft) via file opening dialog (corresponds to the menu item "File/ Open" 69)



Open one of the CFturbo default examples from the installation directory

Open recent project

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.

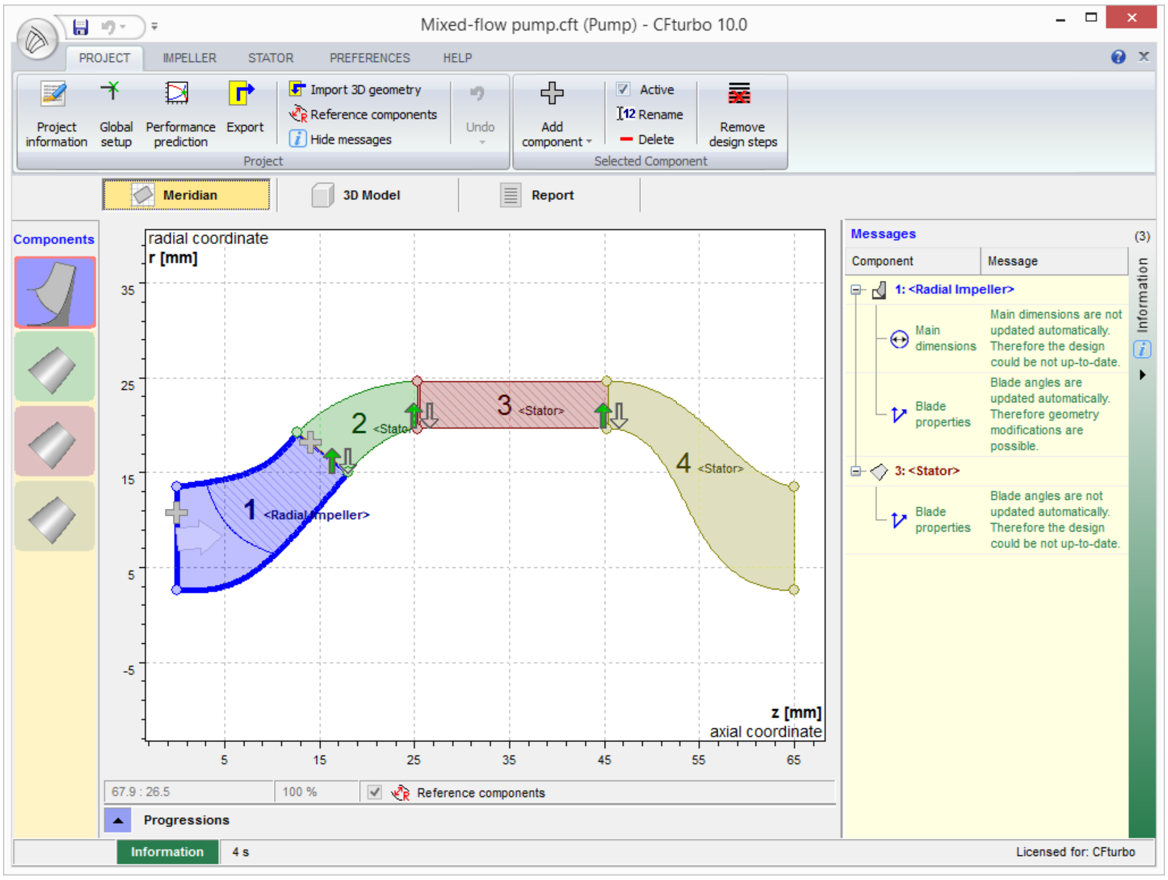


You can clear the entire list using the button right top or use the pop-up menu by right click on any item to remove it or to open the corresponding directory.

Part IV

4 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 65 providing access to all functionality. Some of the ribbon pages are context sensitive.

The CFturbo application window is divided into three main areas:

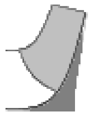
a) Component list on the left side

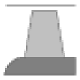
This ordered list contains an icon for each component of the project. The currently selected component is framed.

Clicking on the icon selects the component (alternatively you can click on component in the meridional view 168).

After selecting a component, the ribbon changes to the project tab or to the specific one for this component type (configurable, see General 155). 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)

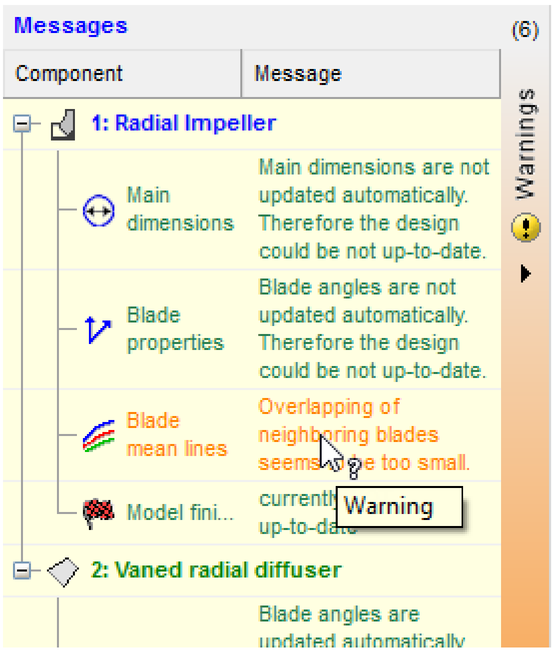

Volute

b) Three alternative views in the central part

see Views 167

c) Message panel on the right side

The message panel shows errors (red), warnings (orange) and information (green) 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 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.

Part V

5 Component design process

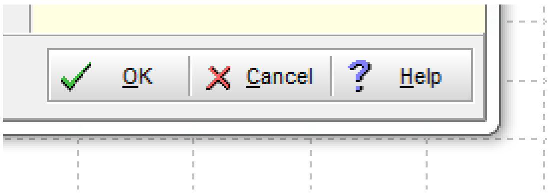
The design process for CFturbo project components requires the completion of a specific sequence of obligatory design steps for each component type (see impeller 189, volute 400, stator 384).

After completing a components basic design process, optional design steps related to model finishing 378 and CFD setup 368 become available.

Each design step comes with its own dialog that can be accessed via the component specific menus 144 or the components context menu 169 in the meridian view.

Design step dialog controls

Generally, dialogs in CFturbo provide the following standard controls:



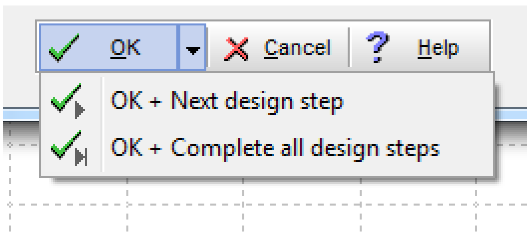
OK
- Closes the dialog and saves user changes into the project.

Cancel
- Closes the dialog and discards all changes made.

Help
- Opens the help topic related to the current design step.

Fast Navigation and Automated component design

Dialogs that are part of the basic component design process provide two more options:



OK + Next design step

- Closes the dialog and opens dialog of the subsequent design step, while saving the user changes into the project.

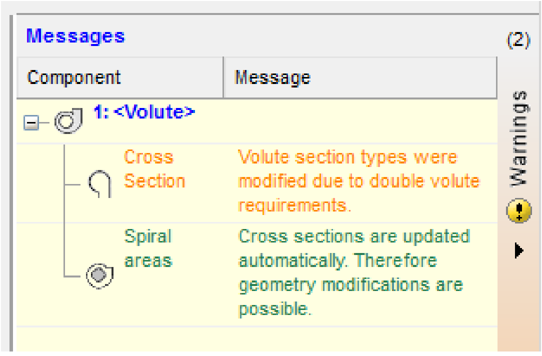
- This feature enables you to quickly navigate all basic design steps in the correct order to apply small modifications faster and more comfortably
- This option is only available, if the selected component has a next design step that is mandatory. Otherwise, it's grayed out.

OK + Complete all design steps

- 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 or has been removed 143 previously. Otherwise, it's grayed out.
- You may use this option as soon as the main dimensions 190 and interfaces 40 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 58 or a message box.

Update Warnings

After any design modification all dependent design steps are updated automatically. In special cases some properties of dependent design steps have to be changed automatically to consider design limitations or to avoid geometrical conflicts. In these cases a message box will be displayed for information:



These information is also displayed in the **Messages** area right in the main form.

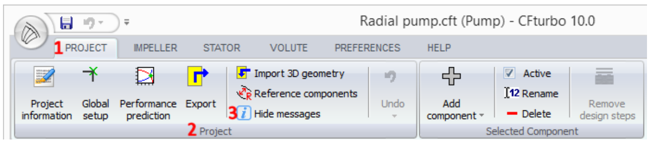
See also Opened project/ message panel 58

Usually you can find more information about a message in the online help by clicking on its text.

Part VI

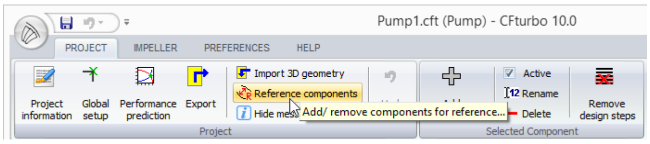
6 Menu

In CFturbo all menus of the main window are located in a ribbon with tabs.

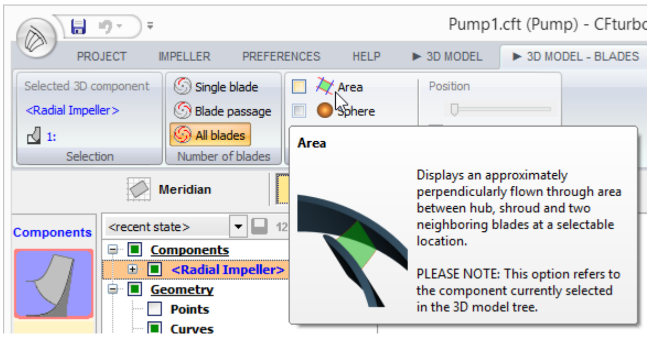


Every tab (**1**) contains groups (**2**) with control elements (**3**).

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



Some buttons have more complex hints, if the function needs more explanation.



By the -Button (CFturbo orb) the file menu 67 can be accessed. Next to it, the quick access toolbar is placed. It can be customized by using the context menu of any element in the ribbon.

The tab pages contain control elements grouped by functionality:

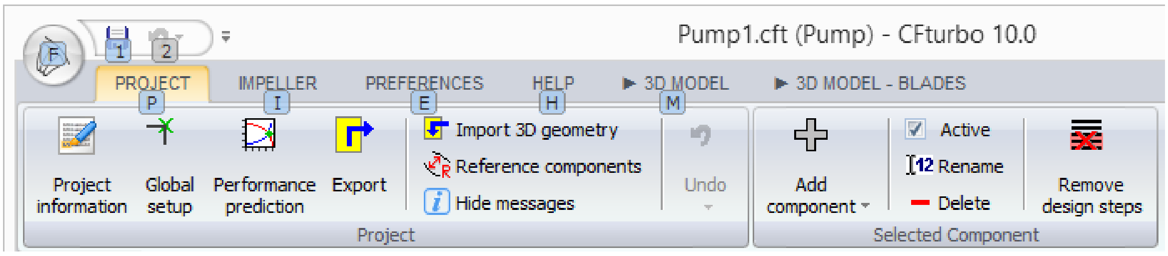
<table>
<tr><th>page</th><th colspan="2">visible</th></tr>
<tr><td>PROJECT 70</td><td colspan="2">if a project is currently opened</td></tr>
<tr><td>IMPELLER 144</td><td rowspan="3">if the current project contains a</td><td>impeller 189</td></tr>
<tr><td>STATOR 144</td><td>stator 384</td></tr>
<tr><td>VOLUTE 144</td><td>volute 400</td></tr>
<tr><td>PREFERENCES 145</td><td colspan="2">always</td></tr>
</table>

HELP 164		
? 3D MODEL 163	if the corresponding view is selected	3D view 172
? 3D-MODEL - BLADES 163		
? REPORT 163		Report view 186

Keyboard shortcuts

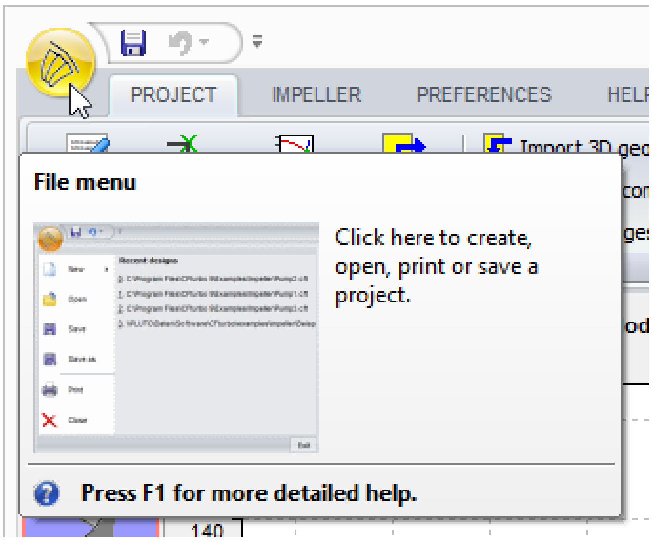
Key tips are displayed, when you press and release the ALT key.

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



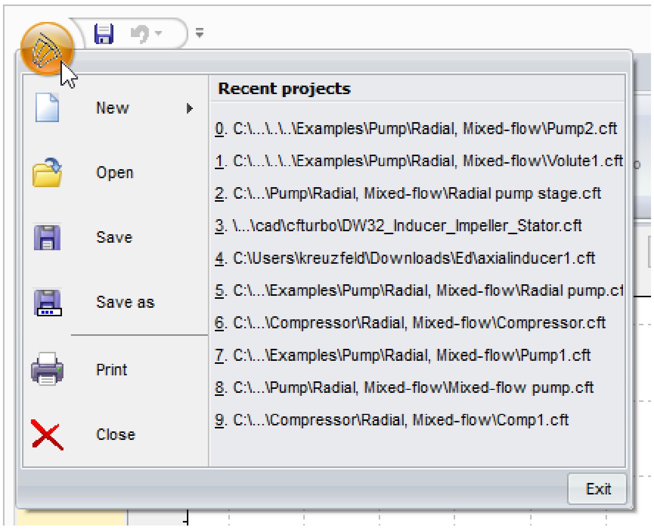
6.1 File

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 54).



6.1.1 Create new design

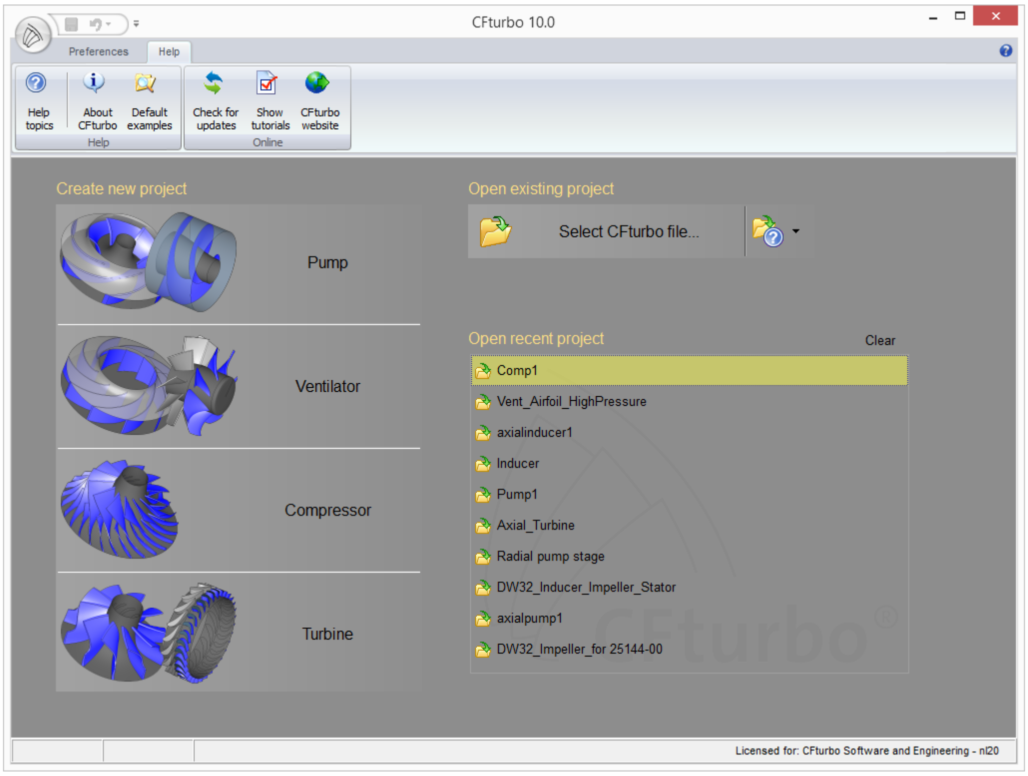
? File | New

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

- Pump

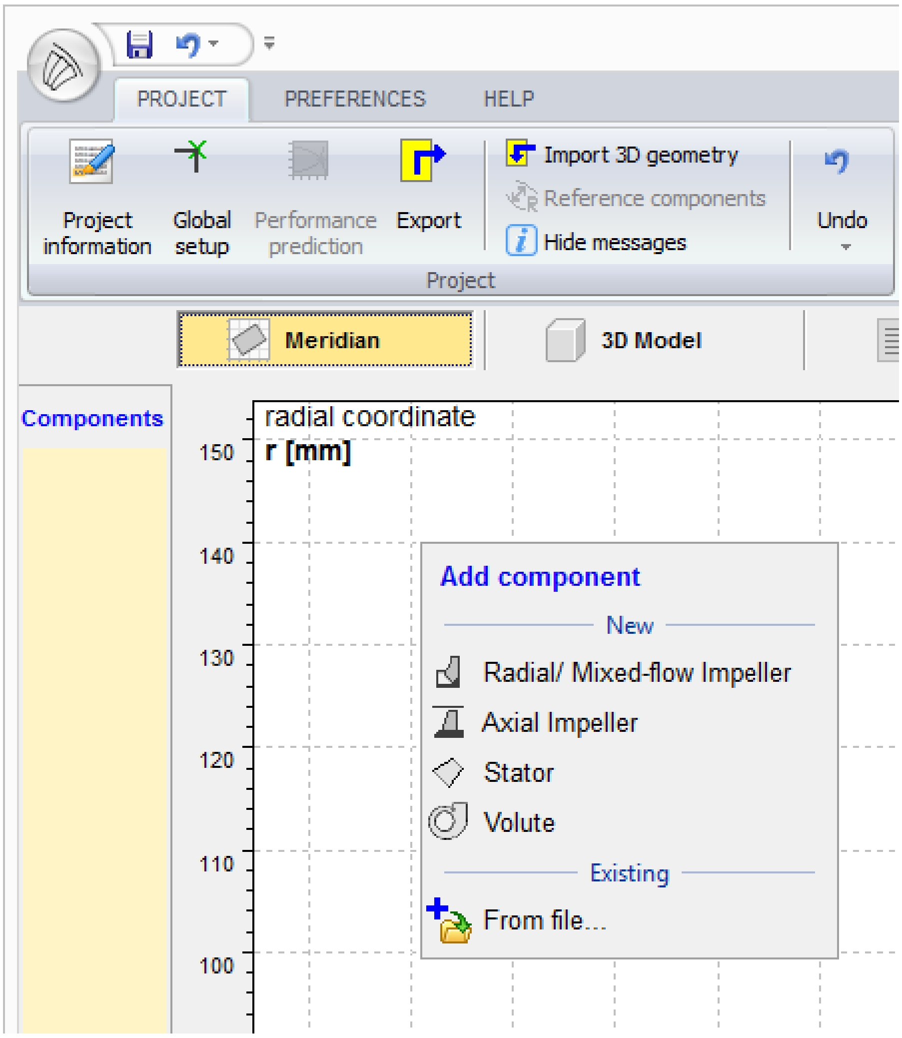
- Ventilator
- Compressor
- Turbine

Equivaltent to using the menu or the toolbar, the buttons in the **Create new project** area can be used, see Start54.



The Global Setup71 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.



6.1.2 Open/ Save design

? File | Open/ Save/ Save as

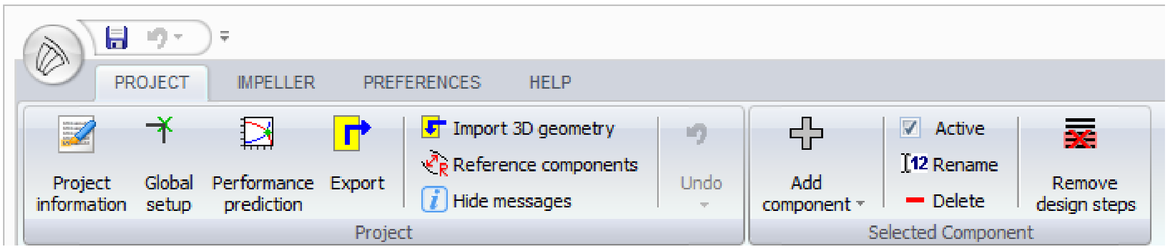
CFturbo projects are saved as *.CFT files (XML file format).

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 **Open Recent Project** if no design is opened, see Start 54.

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

6.2 PROJECT

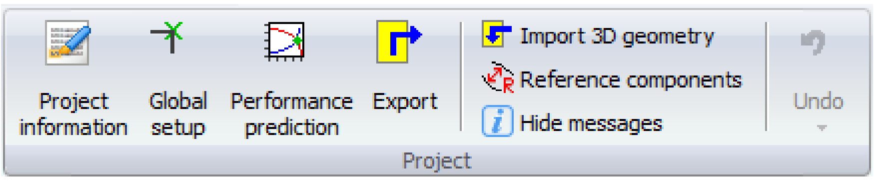
A project can consist of several components (see Project structure and interfaces 38). All components can be designed separately, whereas they influence each other on the interfaces due to geometrical constraints and fluidic coupling.



The Project menu contains those actions, that are related to the whole project (group Project 70) or to the currently selected component (group Selected Component 140).

6.2.1 Project

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



- Project information 71
- Global setup 71
- Performance prediction 77
- Export 85
- Import 3D geometry 135
- Reference components 135
- Show/Hide messages 139
- Undo 139

6.2.1.1 Project information

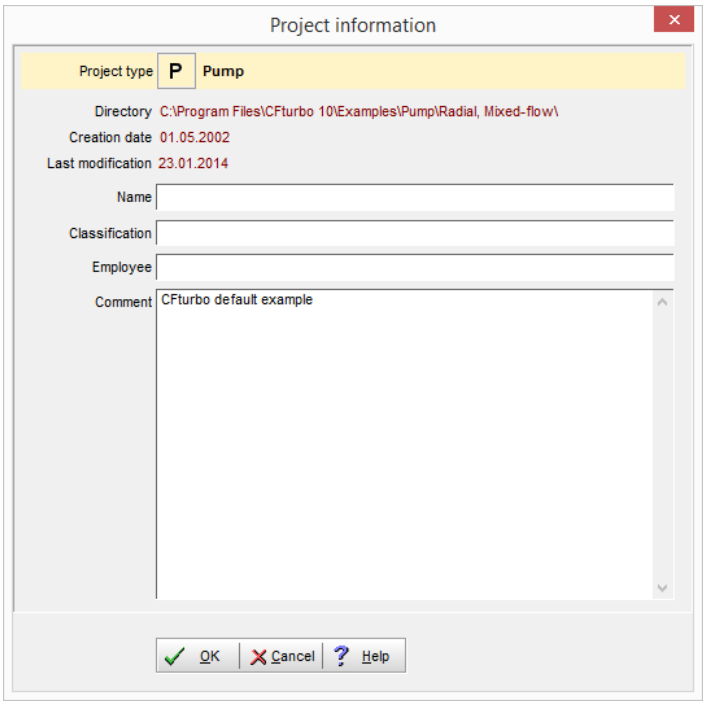
? Project | Project | Project Information

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.



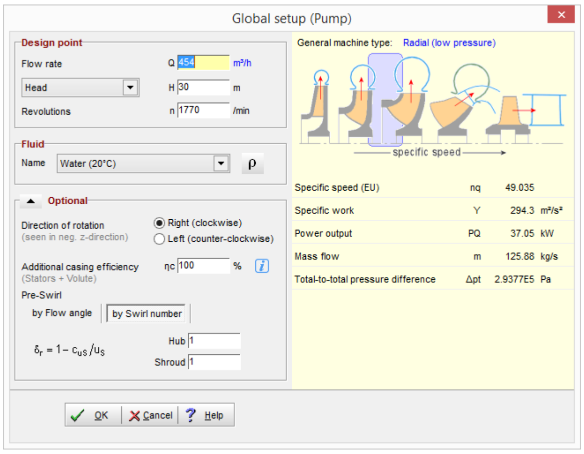
6.2.1.2 Global setup

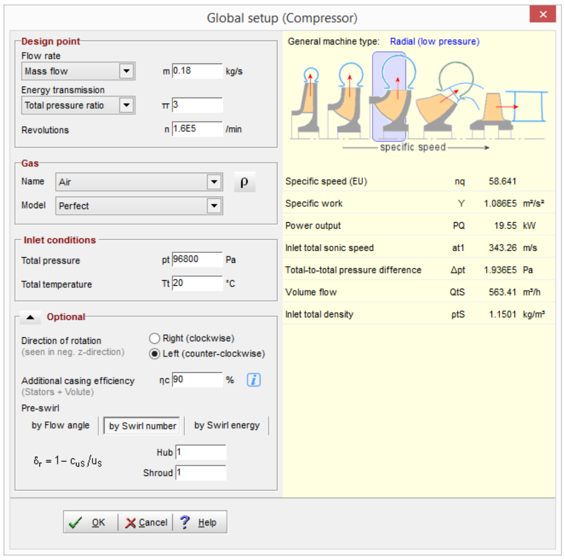
? Project | Project | Global setup

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.





Design point

Here you have to enter the design point data:

(1) Flow rate:

- for pumps, ventilators: volume flow Q
- 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 Δp_t
- for ventilators: total pressure difference Δp_t
- for compressors: total pressure ratio π_t or total pressure difference Δp_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 148.

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

Inlet conditions/ Boundary conditions [for compressors and turbines only]

Here you have to define the total state on suction side by total pressure p_t and total temperature T_t.

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 casing efficiency, which contains all additional (non-typical) flow losses in casing parts of the machine. This efficiency value is used for overall efficiency calculation in addition to the efficiency values specified in the impeller design.
- 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**
	$\alpha_S = \arctan(c_{mS}/c_{uS})$	$\delta_r = 1 - c_{uS}/u_S$	$\delta_Y = u_S c_{uS}/Y$
Positive swirl	$\alpha_S < 90°$	$\delta_r < 1$	$\delta_Y > 0$
Negative swirl	$\alpha_S > 90°$	$\delta_r > 1$	$\delta_Y < 0$
No swirl	$\alpha_S = 90°$	$\delta_r = 1$	$\delta_Y = 0$

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:

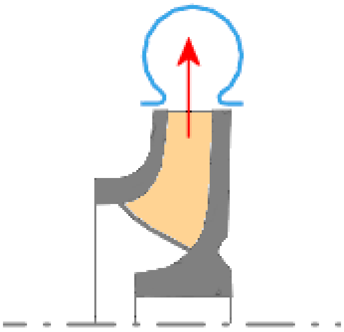
$$\delta_r = 1 - \frac{c_{mS}}{u_S \tan\alpha_S} = 1 - \frac{4Q}{\pi^2 \left(d_S^2 - d_N^2\right) dn \tan\alpha_S}$$

$$\delta_r = 1 - \frac{\delta_Y Y}{u_s^2} \qquad \delta_Y = \frac{u_s^2 (1 - \delta_r)}{Y}$$

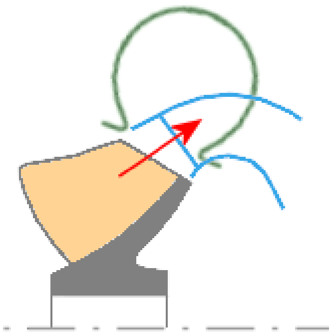
The conversion δ_r - α_S is only valid for certain diameters d_H and d_S.

Information

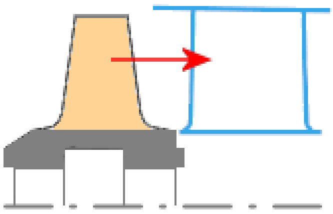
Except for radial-inflow turbines the general meridional shape of the machine, depending on the specific speed, is displayed in the right **Information** area:



radial



mixed-flow
(diagonal)



axial

Furthermore some calculated variables are displayed:

Specific speed	points to machine type and general shape of impeller (see Specific speed 159 definitions)
Specific energy Y	Pumps, Ventilators: $Y = gH = \Delta p_t/\rho$ Compressors (perfect gas model): $Y = \left(\pi_t^{\frac{\kappa-1}{\kappa}} - 1\right) c_p T_{t,S}$
Power output P_Q	$P_Q = \dot{m}Y$ Pumps, Ventilators: $P_Q = \rho gHQ$
Mass flow m	Pumps, Ventilators: $\dot{m} = \rho Q$ Compressors: $\dot{m} = Q_{ts} \cdot \rho_{ts}(p_{ts}, T_{ts})$ (density according to gas model)
Total pressure difference Δp_t	Pumps, Ventilators: $\Delta p_t = \rho gH$ Compressors:

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)	$Q_{ts} = \dfrac{\dot{m}}{\rho_{ts}(p_{ts}, T_{ts})}$ (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)	$T_{t2} = T_{ts}\left(1 + \dfrac{Y}{c_p T_{ts}}\right)$ (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 < nq < 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

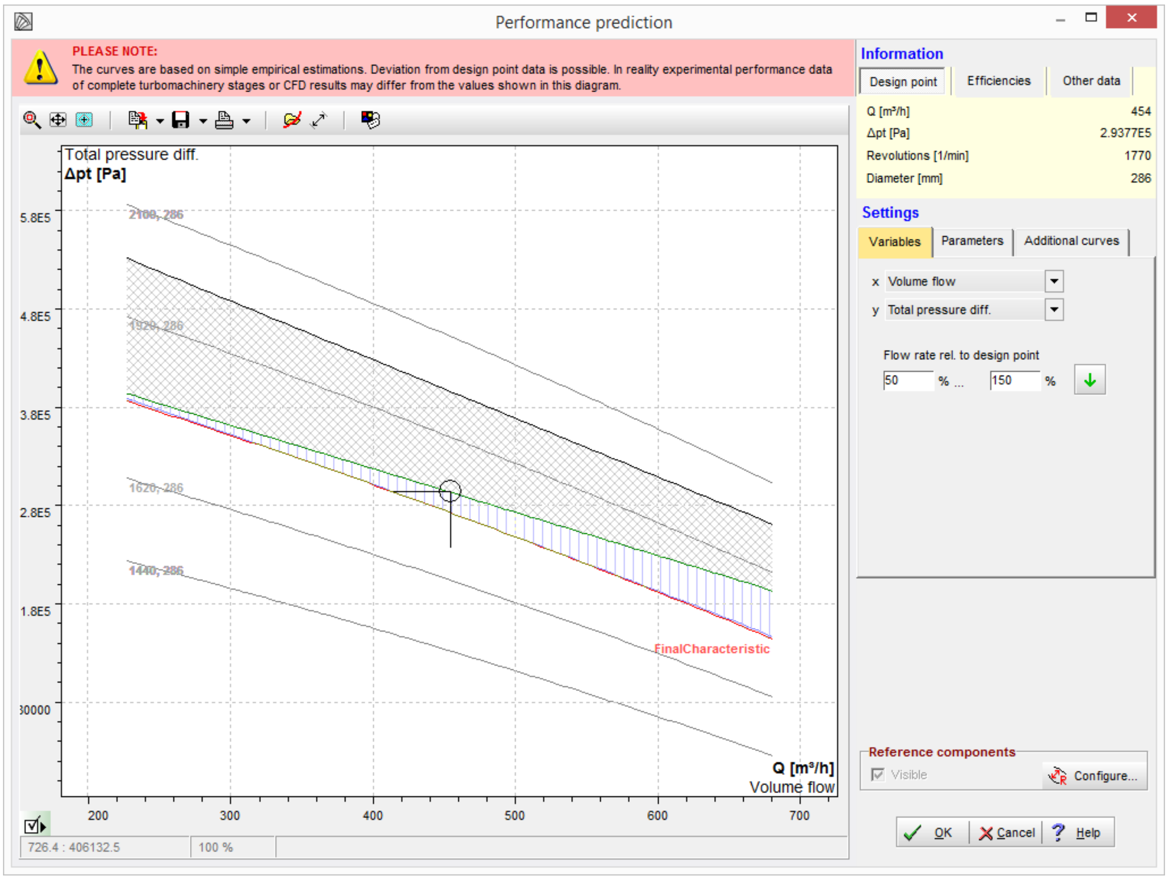
Problem	Possible solutions
Energy transmission of all impellers deviates from globally defined value.	
The sum of energy transmission defined for each impeller deviates from the globally defined value in Global setup.	Check and adapt the energy transmission of the impellers (see Main dimensions 190) to get altogether 100% of the initially defined value of the Global setup.

6.2.1.3 Performance prediction

? Project | Project | 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.

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:

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

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:

.

Kinds of losses

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

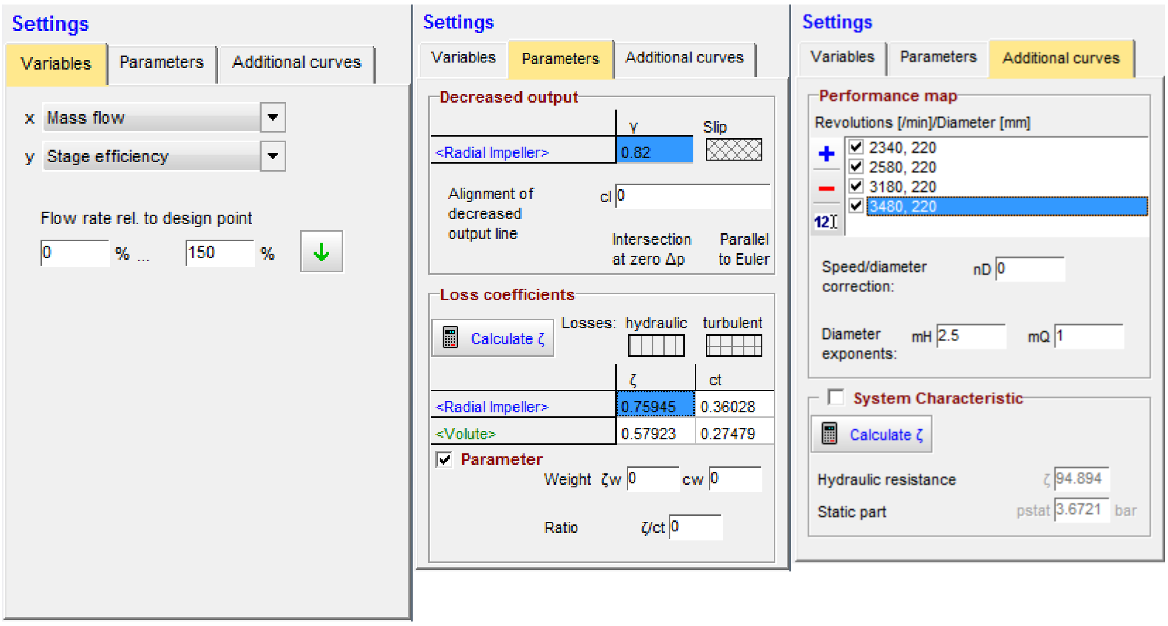
Kind	Description	Parameter	
Decreased power	Based on the Euler-Equation and the decreased power that is calculated in the Blade properties 292. In the design point the decreased power line is shifted by a pressure head loss equivalent to the decreased power ($H_{Decr}=H_{th}-\Delta H_{Decr}$). The decreased power line can be parallel to the Euler-Line as well as positioned that way, that it intersects the Euler-Line at Δp = 0.	cl: cl = 1...parallel position, cl = 0...intersection with Euler-Line at Δp = 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: General approach: $\Delta H_{Hydr} = \zeta$	F: Flow factor that considers the geometry of the component (inlet and outlet area) $F = \frac{100}{g \cdot 0.5 \cdot (A_{in}}$
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.	ct: General approach: $\Delta H_{Turb} = ct$	

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 ζ" 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/ct 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 turbulent losses (ζ, ct)	Additional curves with different speeds and diameter plus system characteristic



The two quadratic approaches towards the description of the hydraulic as well as shock losses (i.e. turbulent 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:

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

Herewith the hydraulic loss become:

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

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

To influence the determination of turbulent losses at $Q < Q_{opt}$ a second weight factor cw is available. With the help of this parameter the turbulent losses become:

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	-	-	-
Δp	total pressure difference			
ψ	work coefficient			
	$\psi = \frac{2 \cdot g \cdot H}{u_2^2}$		$\psi = \frac{2 \cdot Y}{u_2^2}$	$\psi = \frac{2 \cdot Y}{u_1^2}$
H/H_{opt}	head ratio	-	-	-
$\Delta p/\Delta p_{opt}$	total pressure difference ratio			
π_{tt}	-	-	pressure ratio (total-total)	
π_{ts}	-	-	pressure ratio (total-static)	
η_{St}	stage efficiency			
η_{St*}	stage efficiency incl. motor			-
η_V	volumetric efficiency			-
P	required driving power			-
	$P = \frac{\rho \cdot g \cdot H_{Decr}}{\eta_{mech}\eta_{mot}\eta_{sf}} \cdot (Q + Q_{leak}$		$P = \frac{Y_{Decr} \cdot \rho}{\eta_{mech}\eta_{mot}} \cdot (Q + Q_{leak}$	
Q	volume flow			
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 $Q_t = \frac{\dot{m}}{\rho_{t1}}$	$Q_t = \frac{\dot{m}}{\rho_{t2}}$
$\dot{m}$	-	-	mass flow	
$\dot{m}_{red}$	-	-	reduced mass flow $\dot{m}_{red} = \dot{m}\frac{\sqrt{T_{Ref}}}{p_{Ref}}$	
$\dot{m}_{corr}$	-	-	corrected mass flow $\dot{m}_{corr} = \dot{m}\frac{\sqrt{T_{01}/T_{Ref}}}{p_{01}/p_{Ref}}$	

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".

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 exits, that can be found in the table "Kinds of losses", column "Hydraulic losses". Within the applied model the compressor is thought as a parallel connection between a flow source and a hydraulic resistance. Then, surge will occur when the back flow in the hydraulic resistance becomes as big as the flow in the flow source.

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

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

Choke [for compressors only]

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

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

and critical sonic speed becomes:

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

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

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

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 79, last column. The resulting losses will be:

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

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

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

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

Reference curves

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

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.

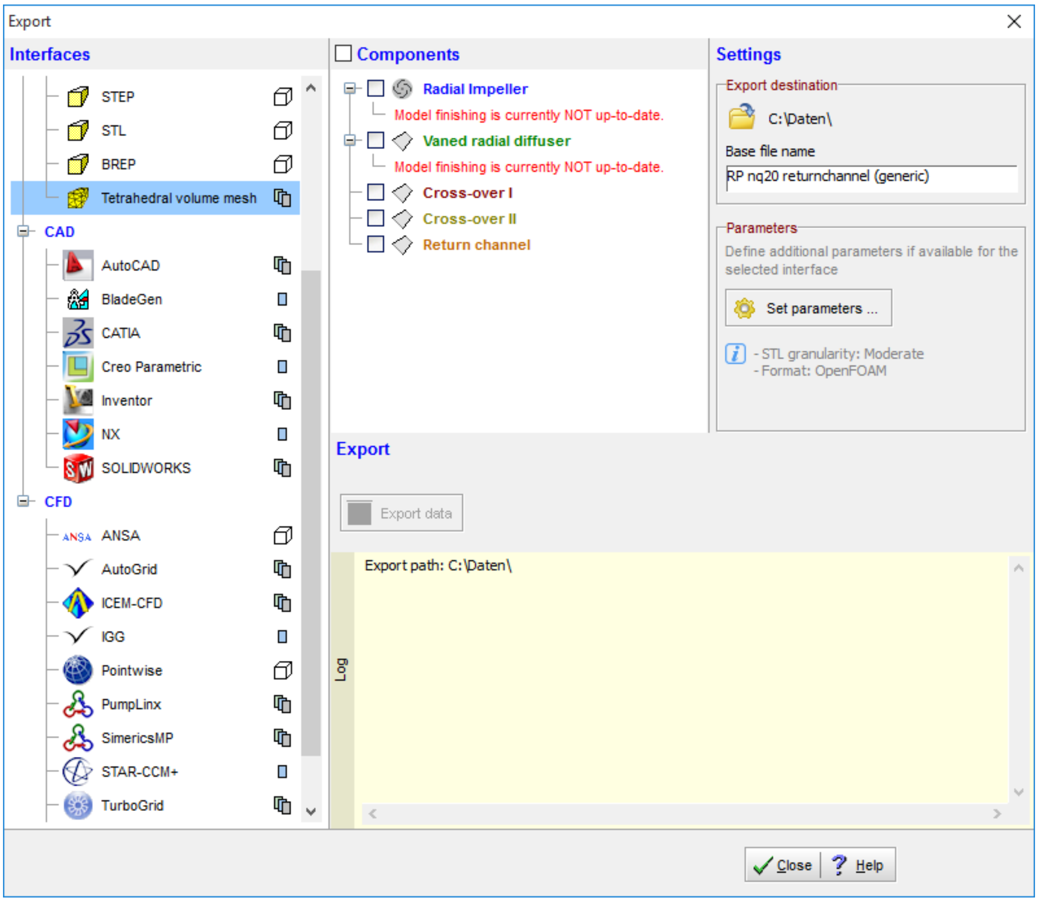
Information

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 is given.

6.2.1.4 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**

Interfaces

Available interfaces are grouped into three blocks: Basic 92, CAD 94 and CFD 97.

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, BREP, ANSA, Pointwise	Tetrahedral volume mesh, ICEM-CFD, PumpLinx, SimericsMP	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 376, Stator 398, Volute 445).

[2)] Point density and export unit can be configured in the **Model settings/ Point export** of each component (Impeller 376, Stator 398, Volute 445).
If the blade shape 292 is ruled surface then points of mean lines as well as profiles (pressure and suction side) are not affected by the model settings 376 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 378.

Components

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

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

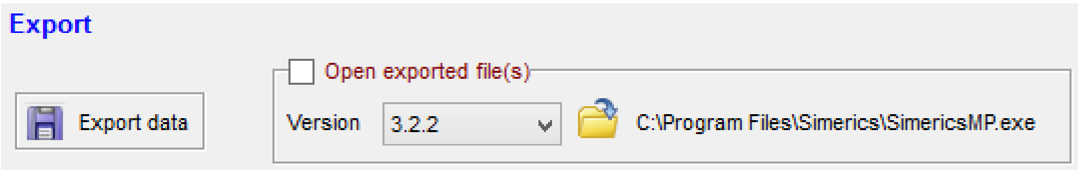
Settings

This area contains all available settings for the selected export interface, like export destination and the base name of exported files. Additional parameters can be available depending on the selected interface.

Export

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

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 368 (generates a segment).
	Blade tip projection to casing required (see CFD Setup).	
	Blade tip projection not accomplished.	Check "Blade projection" in CFD setup/ Miscellaneous 370.
	Gap between leading/ trailing edge and inlet/ outlet required. Select a stator on inlet/ outlet side if possible. Alternatively CFD extension can be activated (see CFD Setup).	
	Some space around blade edges is	Try to increase the distance between

<table>
<tr><th></th><th>Problem</th><th>Possible solutions</th></tr>
<tr><td rowspan="6"></td><td>required for meshing. This can be generated by creating a CFD extension or by selecting a neighbouring stator component.

Note for TurboGrid: a vaneless stator has to be selected, which has to be considered as part of the rotating domain in TurboGrid.</td><td>leading/ trailing edge and meridional inlet/ outlet by

a) moving leading/ trailing edge in meridional contour if edge is not fixed on inlet/ outlet 270.

b) selecting a neighbouring stator if possible.

or

c) activating of CFD-Extension in CFD setup/ Extension 368.</td></tr>
<tr><td colspan="2">Gap between leading/ trailing edge and inlet/ outlet recommended. CFD extension can be activated (see CFD Setup).</td></tr>
<tr><td>Some space around blade edges is recommended.</td><td>Try to increase the distance between leading/ trailing edge and meridional inlet/ outlet by

a) moving leading/ trailing edge in meridional contour if edge is not fixed on inlet/ outlet 270.

or

b) activating of CFD-Extension in CFD setup/ Extension 368.</td></tr>
<tr><td colspan="2">Small gap between blade/ leading edge and inlet/ outlet could cause import problems. Try to increase it if you experience any problems on import.</td></tr>
<tr><td>See message.</td><td>Try to increase the distance between leading/ trailing edge and meridional inlet/ outlet by

a) moving leading/ trailing edge in meridional contour if edge is not fixed on inlet/ outlet 270.

or

b) activating of CFD-Extension in CFD setup/ Extension 368 (only for impellers).</td></tr>
<tr><td colspan="2"></td></tr>
<tr><td rowspan="2">Finishing</td><td colspan="2">Trimmed solid is required (see Model finishing).</td></tr>
<tr><td>Up-to-date trimmed solids required.</td><td>Execute Model finishing 378 with option "Solid trimming".</td></tr>
</table>

	Problem	Possible solutions
	Extended blade (see Model finishing) not supported.	
	See message.	Execute Model finishing 378 with option "No model finishing" or "Solid trimming".
	Model finishing is currently NOT up-to-date.	
	See message.	Execute Model finishing 378.
	No model finishing is currently selected. "Solid trimming" is recommended for surface/ solid export.	
	See message.	Execute Model finishing 378 with option "Solid trimming".
	Trimmed solid (incl. features like fillets) not supported by point based export formats.	
	See message.	-
	Fillet-Cut-water is not supported by point based export formats. Cutwater has to be designed manually in CAE.	
	See message.	-
	Solid vs. Solid faces: They are handled differently by various target systems.	
	To be taken into account if a mixed selection of solids and solid faces was selected in the component tree 179.	-
	Export of "Flow Domain" might be defective.	
	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 component tree 179.
Blades	**Complete blade edges design is required.**	
	"Blade edges" design step not accomplished.	Execute Blade edges 344.
	Blades are required (see Main dimensions).	
	Components without blades are no	-

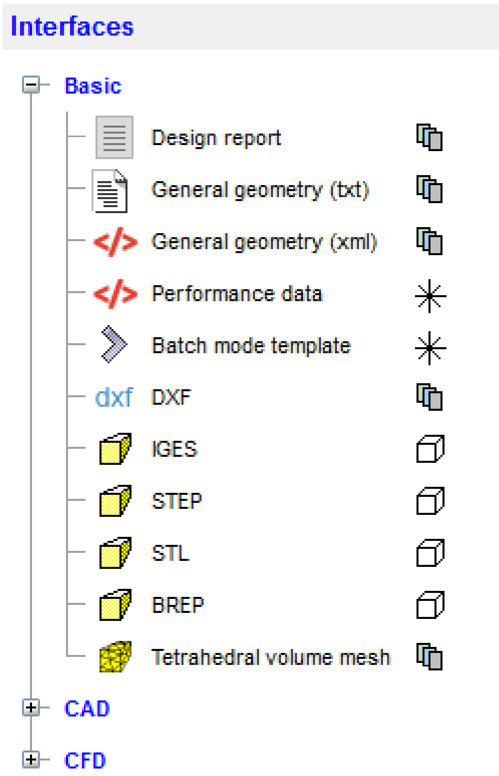
	Problem	Possible solutions
	supported by this interface.	
	Designs with only one blade are not supported.	
	See message.	-
	Designs with blade wrap angle larger than 360° are not supported.	
	See message.	-
	RTZT format does not support blades with asymmetric thickness distribution.	
	Blades with asymmetric thickness distribution will be imported in BladeGen, so that the thickness distribution is symmetric with respect to the mean line.	-
Model settings	**Geometry is not fully included in a cube between (-500,-500,-500) and (500,500,500). Choose other export units.**	
	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 the Parasolid™ library limitation.	Change length unit in Model settings/ Point export 376.
	Current point export settings could cause import problems in Inventor due to high number of points.	
	See message.	Change number of points in Model settings/Point export 376.
	Different export units were selected for at least two selected components.	
	See message.	Select identic export units for all components in Model settings/Point export 376.
General	**Complete all design steps is required.**	
	Only for CFD-Applications. One or more design steps were not finished.	Complete all design steps.
	Special license for this interface required.	

	Problem	Possible solutions
	License for this interface not found.	Check the license information in Preferences/Licensing 145.
	No license available.	
	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.
	Parts to be exported have to be visible in the 3D Model. Imports can only be exported via the context menu of the 3D Model tree.	
	See message.	Make all parts to be exported visible in the component tree 179.
	Performance prediction not supported for axial turbine projects.	
	See message.	-
	Performance prediction not supported for projects without any impellers.	
	See message.	-
	Volutes without cut-water are not supported.	
	CFturbo2ICEM does not support volutes without cut-water.	-
	Invalid viscosity value.	
	See message.	Set a valid viscosity value in fluid manager 148.
	Real gas properties will be ignored by default. You have to configure the *.rgp file manually if required.	
	Only for Vista TF. See message.	-
	Parts of an inactive component are visible in the 3D Model. They will not be exported.	
	See message.	Make all visible parts for inactive components invisible in the component tree 179.

6.2.1.4.1 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. The formats IGES, STEP, STL and BREP export the geometry visible in the 3D model.

[**I** = **I**mpeller **S** = **S**tator **V** = **V**olute **MC** = **M**ulti-**C**omponent 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					

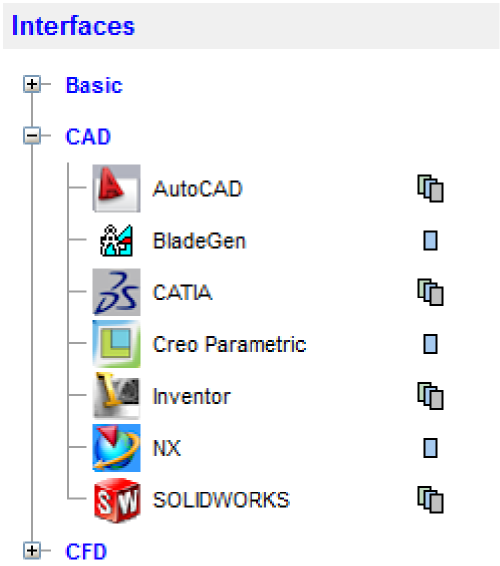
	see Report 186					
General geometry (txt)	*.cft-geo	general text file	I	S	V	MC
	Text file containing geometry data of the design for any further processing. **Impeller/stator:** Meridional section: z, r of hub, shroud, leading edge Blade mean lines, Blade profiles: x, y, z: cartesian coordinates r: radius t: angle T: tangential length m: meridional radius based length m/m_{TE}: meridional radius based length (0..1) M: meridional absolute length M/M_{TE}: meridional absolute length (0..1) β: blade angle s: blade thickness L: 3D length la: lean angle **Volute:** Spiral cross sections, Diffuser cross sections: x, y, z (cartesian coordinates) Contour lines in circumferential direction: x, y (cartesian coordinates)					
General geometry (xml)	*.cft-geo-xml	general xml file	I	S	V	MC
	XML file containing geometry data of the design for any further processing.					
Performance data	*.cft-pp	XML format file	File is created for whole project			
	XML file containing results of Performance prediction 77					
Batch mode template	*.cft-batch	XML format file	File is created for whole project			
	It contains all changeable values with a short description and sample actions. see Batch mode 26					
DXF	*.dxf	neutral format (Drawing Interchange File Format)	I	S	V	MC
	File contains designed geometry of the selected component as 3D polylines.					
IGES	*.igs	neutral format (Initial Graphics Exchange Specification)	Components and elements are			

<table>
<tr><td></td><td colspan="2">File contains designed geometry as 3D surfaces. Visible 3D view is the basis.</td><td colspan="4" rowspan="7">selected in 3D view</td></tr>
<tr><td rowspan="2">STEP</td><td>*.stp</td><td>neutral format
(Standard for the Exchange of Product model data)</td></tr>
<tr><td colspan="2">File contains designed geometry as 3D surfaces. Visible 3D view is the basis. Also, the names displayed in the model tree are exported.
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 STAR-CCM+, it is better to export solids instead of solid faces.
For SOLIDWORKS, try with and without STEP import option: "B-REP mapping".</td></tr>
<tr><td rowspan="2">STL</td><td>*.stl</td><td>neutral format
(Standard Triangulation Language)</td></tr>
<tr><td colspan="2">File contains designed geometry as triangulated 3D surfaces. Some parameters 100 can be adapted. Visible 3D view is the basis.</td></tr>
<tr><td rowspan="2">BREP</td><td>*.brep</td><td>native format of Open CASCADE based applications
(Boundary Representation)</td></tr>
<tr><td colspan="2">File contains designed geometry as 3D surfaces. Visible 3D view is the basis.</td></tr>
<tr><td rowspan="2">Tetrahedral volume mesh</td><td>*.msh, *.vol, polyMesh</td><td>3 alternative file formats are available:
Fluent, Netgen, OpenFOAM</td><td>I</td><td>S</td><td>V</td><td>MC</td></tr>
<tr><td colspan="2">File contains designed geometry as tetrahedral volume mesh for simulation. File format and mesh resolution can be specified with Set parameters.</td><td></td><td></td><td></td><td></td></tr>
</table>

6.2.1.4.2 CAD

? Project | Export | CAD

The CAD group contains the supported CAD product interfaces.



Export preconditions

The export availability of CAD 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"

The interfaces AutoCAD, CATIA, Inventor and SOLIDWORKS support multi-component export.

[**I** = **I**mpeller **S** = **S**tator **V** = **V**olute **MC** = **M**ulti-**C**omponent export supported]

Menu entry	Description		Component type			
AutoCAD	*-.txt	Version 2014	**I**	**S**	**V**	**MC**
	Lisp script *xyz2spline* (part of CFturbo) creates splines from imported points. • Select "AutoCAD Classic" Workspace • Load "xyz2spline.lsp" under Manage | Load Application • Run command "xyz2spline" and select *.txt file					
BladeGen	*.rtzt	Version 14.5, 15	**I**	**S**	**V**	**MC**

<table>
<tr><td></td><td colspan="2">The file contains complete 3D impeller geometry point-by-point.
• File | Open: select file type „Meanline File (*.rtzt)“
• select *.rtzt file</td><td></td><td></td><td></td><td></td></tr>
<tr><td rowspan="2">CATIA</td><td>*.catvbs</td><td>Version V5R19</td><td>I</td><td>S</td><td>V</td><td>MC</td></tr>
<tr><td colspan="2">The macro generates a surface model + generating splines.
• Tools | Macro | Macros
• Select macro library and macro, Run</td><td></td><td></td><td></td><td></td></tr>
<tr><td rowspan="2">Creo Parametric</td><td>*.ibl, *.pts</td><td>Version 2.0 M090</td><td>I</td><td>S</td><td>V</td><td>MC</td></tr>
<tr><td colspan="2">*.ibl contains geometry defined by 3D points.
*.pts files are exported for impellers only and contain information about blade thickness defined by 2D points
• Home | New | Part <name> (if no file is open)
• Model | Get data | Import
• select *.ibl or *.pts file</td><td></td><td></td><td></td><td></td></tr>
<tr><td rowspan="2">Inventor</td><td>*.bas</td><td>Version 2014</td><td>I</td><td>S</td><td>V</td><td>MC</td></tr>
<tr><td colspan="2">The macro generates a surface model + generating splines.
• Tools | Visual Basic Editor
• VB
o File | New project
o File | Import file, select *.bas
o Tools | Macro, select “Main”, Run</td><td></td><td></td><td></td><td></td></tr>
<tr><td rowspan="2">NX</td><td>*-ug.dat</td><td>Version 8.0</td><td>I</td><td>S</td><td>V</td><td>MC</td></tr>
<tr><td colspan="2">One file per component will be created.
• New | New Project file <name> (if no file is open)
To import curves (hub, shroud, volute contour curves):
• Insert | Curve | Spline | Through points
• Points from file
• select *.dat file
To generate surfaces (blade, volute, diffuser):
• Insert | surface | Through points
• Row degree <= number of blade profile sections
• Column degree <= Row degree-1
• Points from file
• select *.dat file

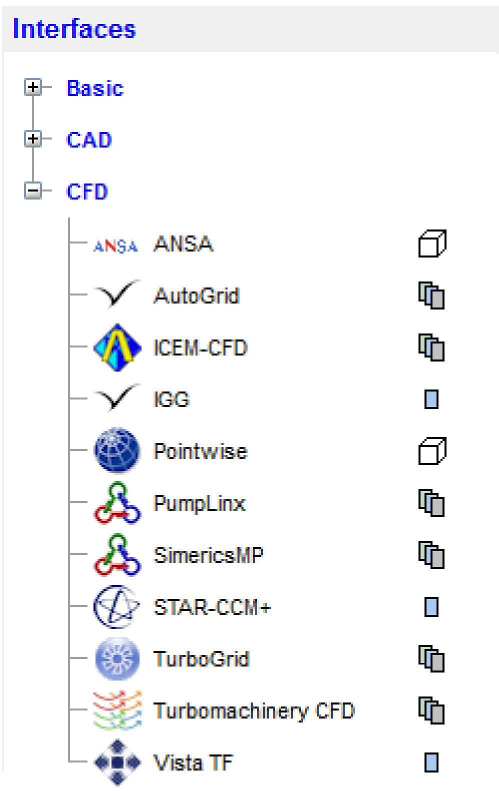
Please note: If the mentioned menu options are not available, the appropriate commands have to be created:
a) "Tools/Customize" or right click on any toolbar/menu, "Customize..."
b) "Commands", "Insert/Curve/Spline..." or "Insert/Surface/</td><td></td><td></td><td></td><td></td></tr>
</table>

<table>
<tr><td></td><td colspan="2">Through Points..."
c) Integrate selected item via Drag and Drop in a menu or toolbar</td><td></td><td></td><td></td><td></td></tr>
<tr><td>SOLIDWORKS</td><td>*.swb</td><td>Version 2014</td><td>I</td><td>S</td><td>V</td><td>MC</td></tr>
<tr><td></td><td colspan="2">The macro generates a surface model + generating splines.
• Tools | Macro | Run: select *.swb</td><td></td><td></td><td></td><td></td></tr>
</table>

6.2.1.4.3 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, Pointwise, PumpLinx and Simerics MP support multi-component export.

[**I** = **I**mpeller **S** = **S**tator **V** = **V**olute **MC** = **M**ulti-**C**omponent export supported]

<table>
<tr><th>Menu entry</th><th colspan="2">Description</th><th colspan="4">Component type</th></tr>
<tr><td rowspan="2">ANSA</td><td>*.igs</td><td>Version 15.3</td><td>I</td><td>S</td><td>V</td><td>MC</td></tr>
<tr><td colspan="2">• File | Open
• Select *.igs file</td><td></td><td></td><td></td><td></td></tr>
<tr><td rowspan="2">AutoGrid</td><td>*.geomTurbo</td><td>Version 9.1.3</td><td>I</td><td>S</td><td>V</td><td>MC</td></tr>
<tr><td colspan="2">• File | New Project
• “Initialize a New Project from a geomTurbo File”
• Select *.geomTurbo file</td><td></td><td></td><td></td><td></td></tr>
<tr><td rowspan="2">ICEM-CFD</td><td>*.tinXML, *.stp</td><td>Version 13, 14, 14.5, 15</td><td>I</td><td>S</td><td>V</td><td>MC</td></tr>
<tr><td colspan="2">A STEP file with named geometries is created. The names are visible in ICEM-CFD if the file is imported via Workbench Reader.

Parameters are saved in a separate XML file.</td><td></td><td></td><td></td><td></td></tr>
<tr><td rowspan="2">IGG</td><td>*.dat</td><td>Version 9.1-3</td><td>I</td><td>S</td><td>V</td><td>MC</td></tr>
<tr><td colspan="2">Multiple data files are generated: section.dat, diffusor.dat, curves.dat
• File | Import | IGG Data
• Select *.dat file
• Repeat steps for remaining files</td><td></td><td></td><td></td><td></td></tr>
<tr><td rowspan="2">Pointwise</td><td>*.igs</td><td>Version 17.0R2</td><td>I</td><td>S</td><td>V</td><td>MC</td></tr>
<tr><td colspan="2">• File | Import | Database
• Select *.igs file</td><td></td><td></td><td></td><td></td></tr>
<tr><td rowspan="2">PumpLinx
Simerics MP</td><td>*.spro, *.stl</td><td>Version 3.4.9</td><td>I</td><td>S</td><td>V</td><td>MC</td></tr>
<tr><td colspan="2">The *.spro file contains all project information. The *.stl file contains the geometry in STL format as triangulated 3D surfaces. Some parameters 100 can be adapted.

In Simerics MP/ PumpLinx: Select *.spro file under Open project</td><td></td><td></td><td></td><td></td></tr>
</table>

STAR-CCM+	*.bndy, *.estg, *.trbw	Version 8.04.007	**I**	**S**	**V**	**MC**
	• Mesh | Import turbo blades... • Select *.trbw file under Load Turbo Wizard Settings					
TurboGrid	*.curve	Version 14.5	**I**	**S**	**V**	**MC**
	4 files are created, a session file (<filename>.tse) and <filename>_hub.curve, <filename>_shroud.curve, <filename>_profile.curve. Load the saved session file <filename>.tse: • File | New Case • Session | Play Session or Open the curve files (<filename>_hub.curve, <filename>_shroud.curve, <filename>_profile.curve) manually: • 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					
Vista TF	*.fil, *.con, *.geo, *.aer, *.cor	Version 4.05	**I**	**S**	**V**	**MC**
	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.					

ICEM-CFD (ANSYS)

This interface is supporting the script solution CFturbo2ICEM for automated meshing of CFturbo geometries. Detailed information can be found on the CFturbo website.

The button **Set parameters...** opens the **Export ICEM-CFD** 131 dialog for defining meshing parameters. These settings are saved in the *.tinXML file, whereas the geometry is transferred by a *.stp file.

For more information about using CFturbo2ICEM please see the available documentation.

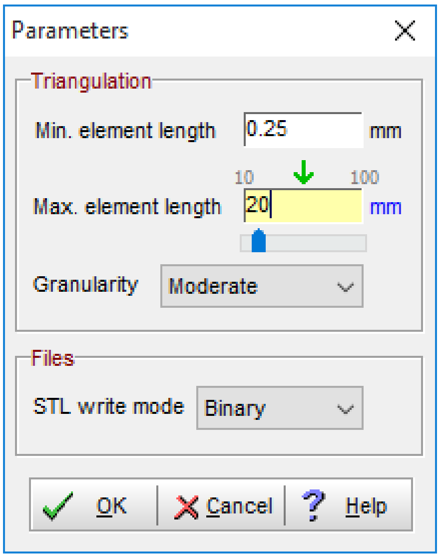
6.2.1.4.4 Specifics

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

- AutoCAD (Autodesk, Inc.) 101
- Inventor (Autodesk, Inc.) 125
- CATIA (Dassault Systèmes) 108
- AutoGrid (NUMECA International) 128
- Creo Parametric (PTC, Inc.) 109
- ICEM-CFD (ANSYS) 131
- STL 100

6.2.1.4.4.1 STL

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



Minimum element length: Minimum mesh element length.

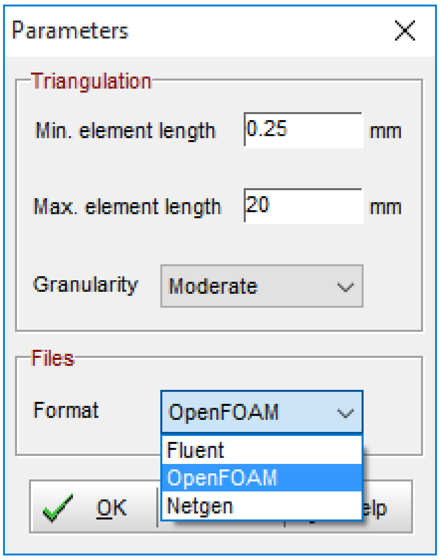
Maximum element length: Maximum mesh element length.

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.

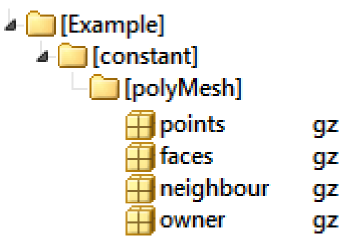
6.2.1.4.4.2 Tetrahedral volume mesh

In addition to the parameters[100] for triangulation, 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

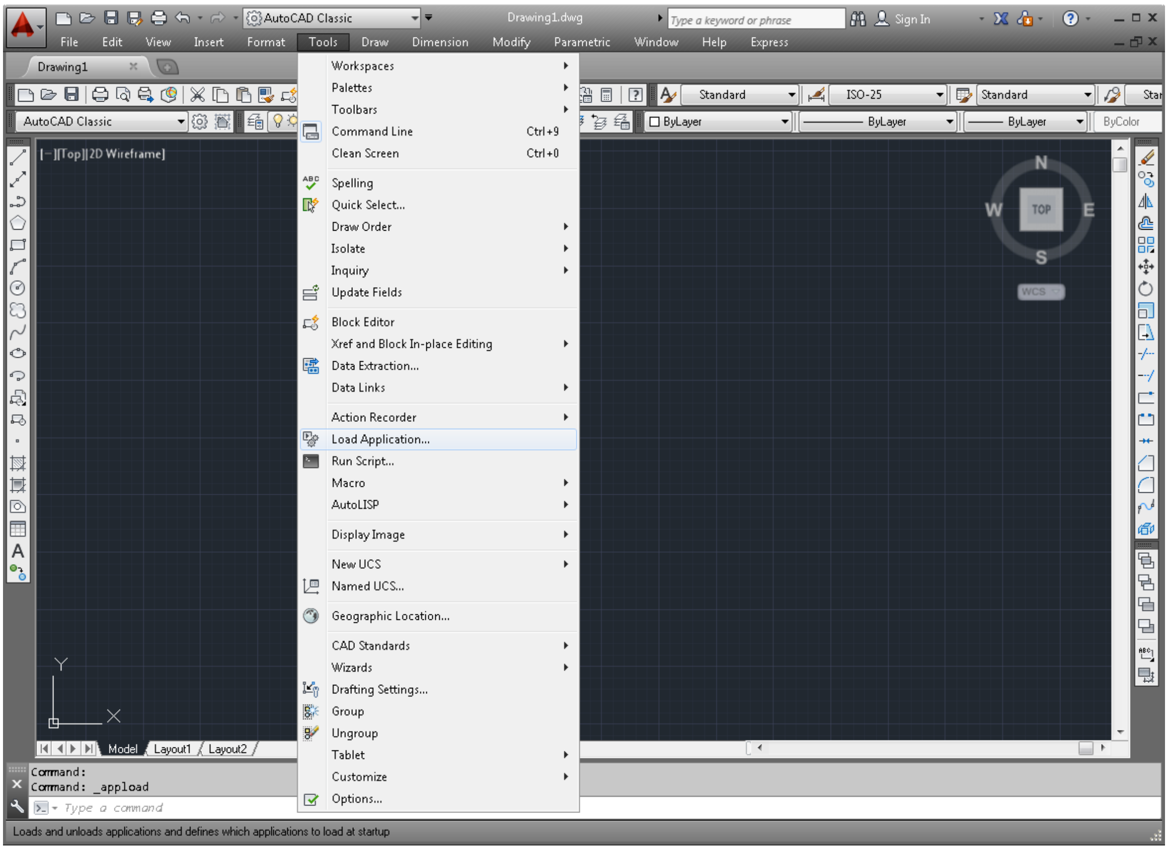
6.2.1.4.4.3 AutoCAD (Autodesk, Inc.)

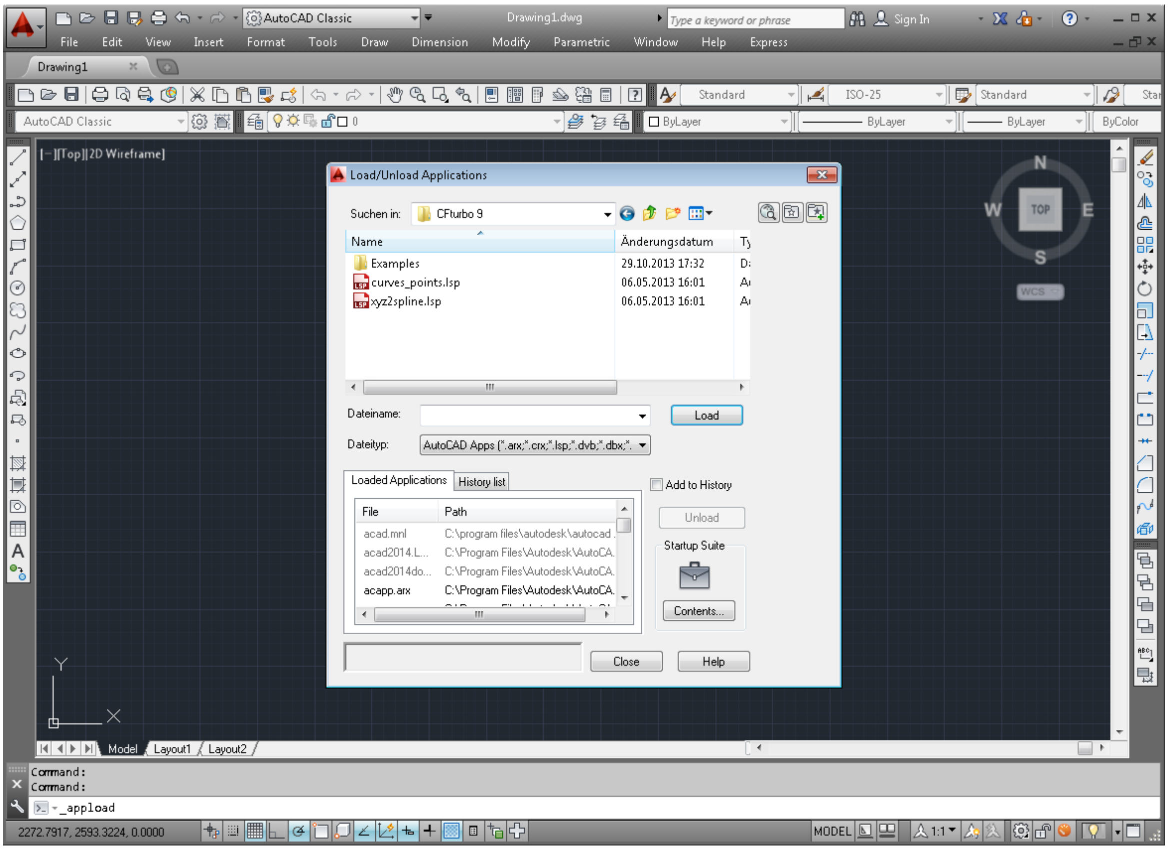
The data import from CFturbo is realized by a LISP-script.

Loading the LISP-Application and Import of the Geometry

- Select "AutoCAD Classic" Workspace
- Manage | 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.



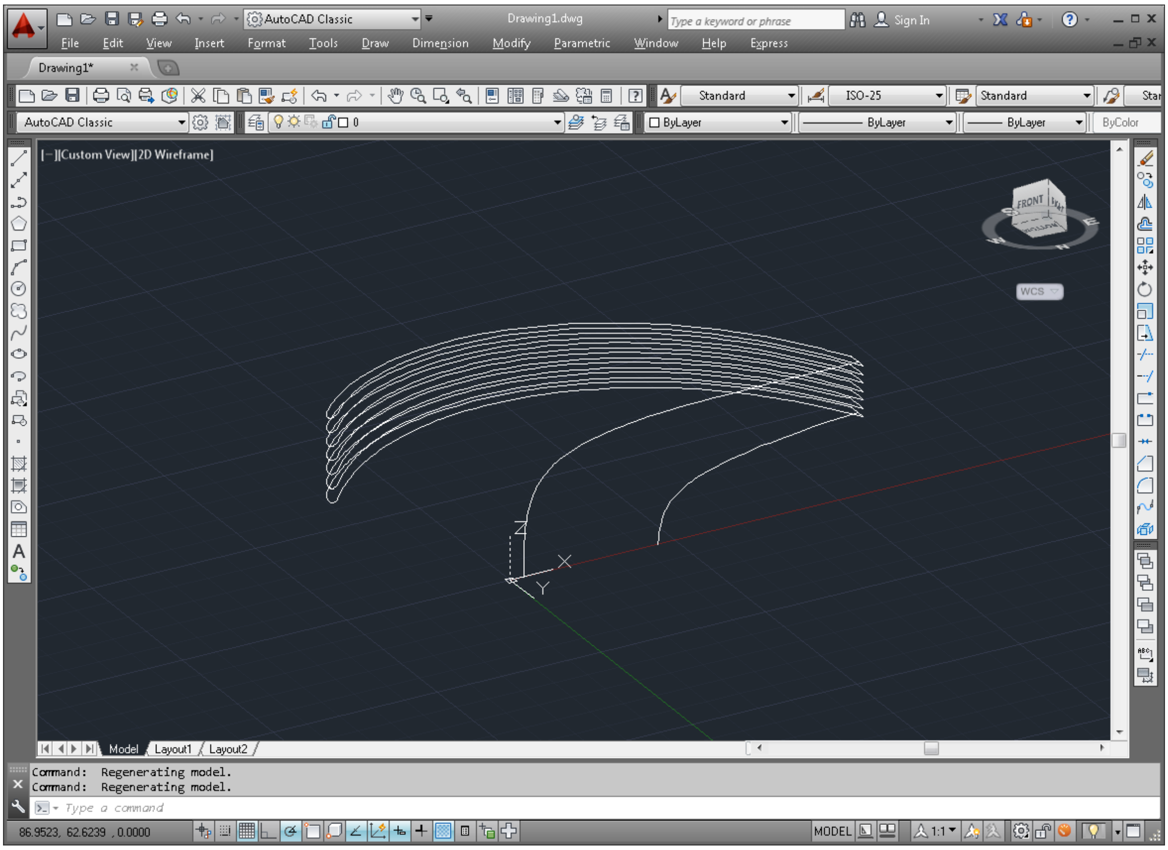


Selection of xyz2spline.lsp file

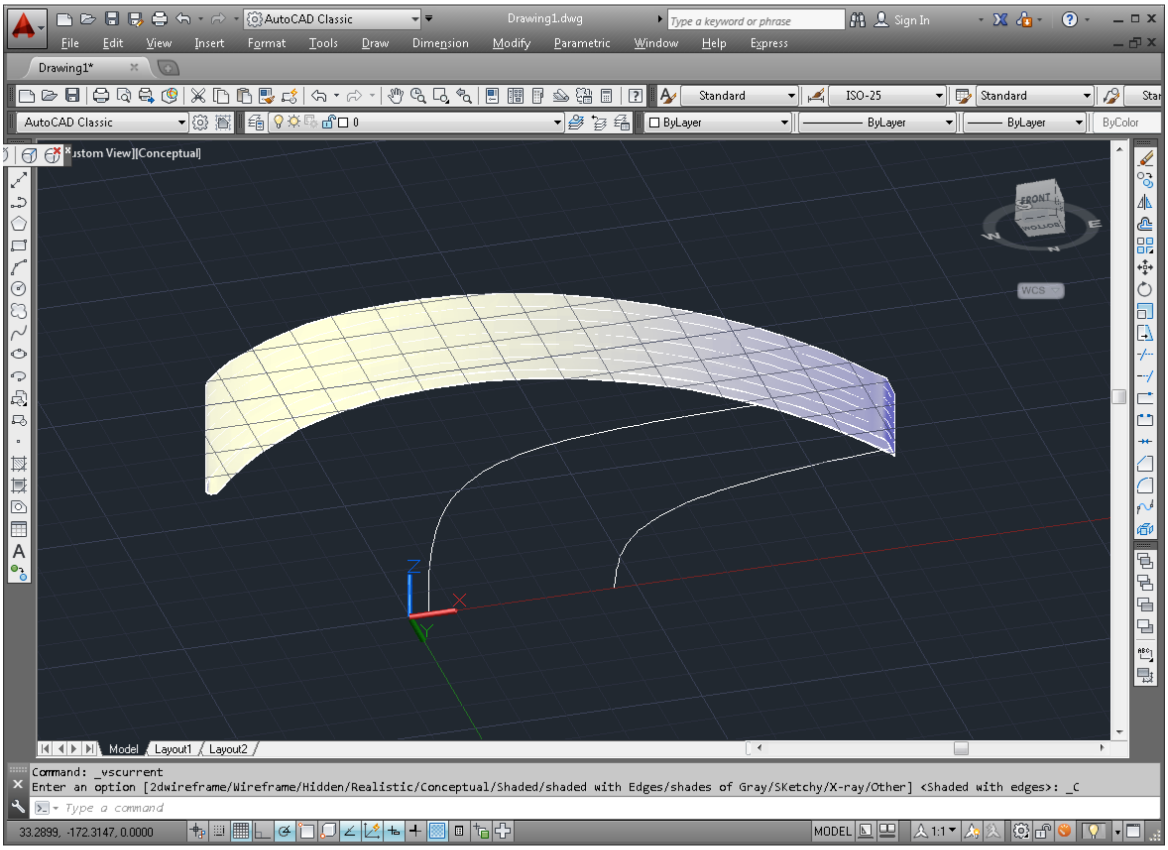
Construction of Impeller

Creating the blades

- Use the command _*loft* to create surfaces from curves



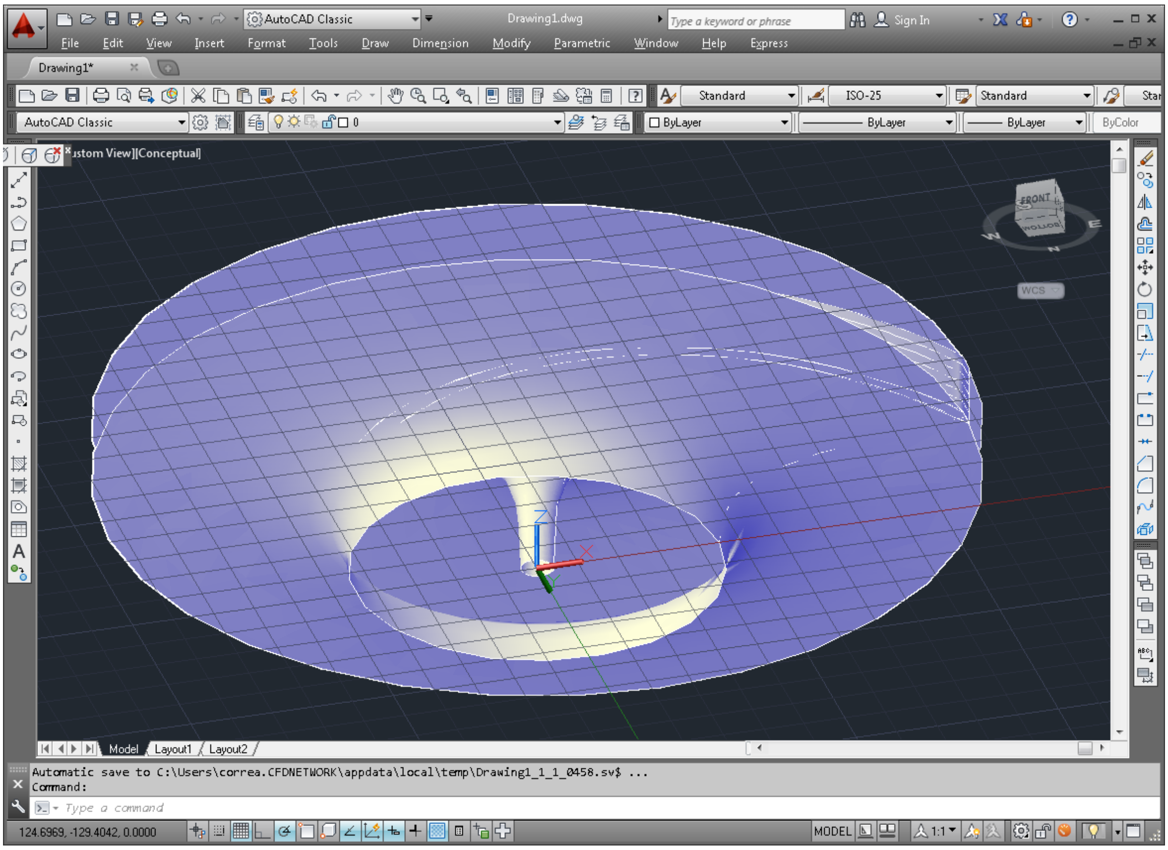
Sample-view after data import



Blade surface gerated by using the _loft command

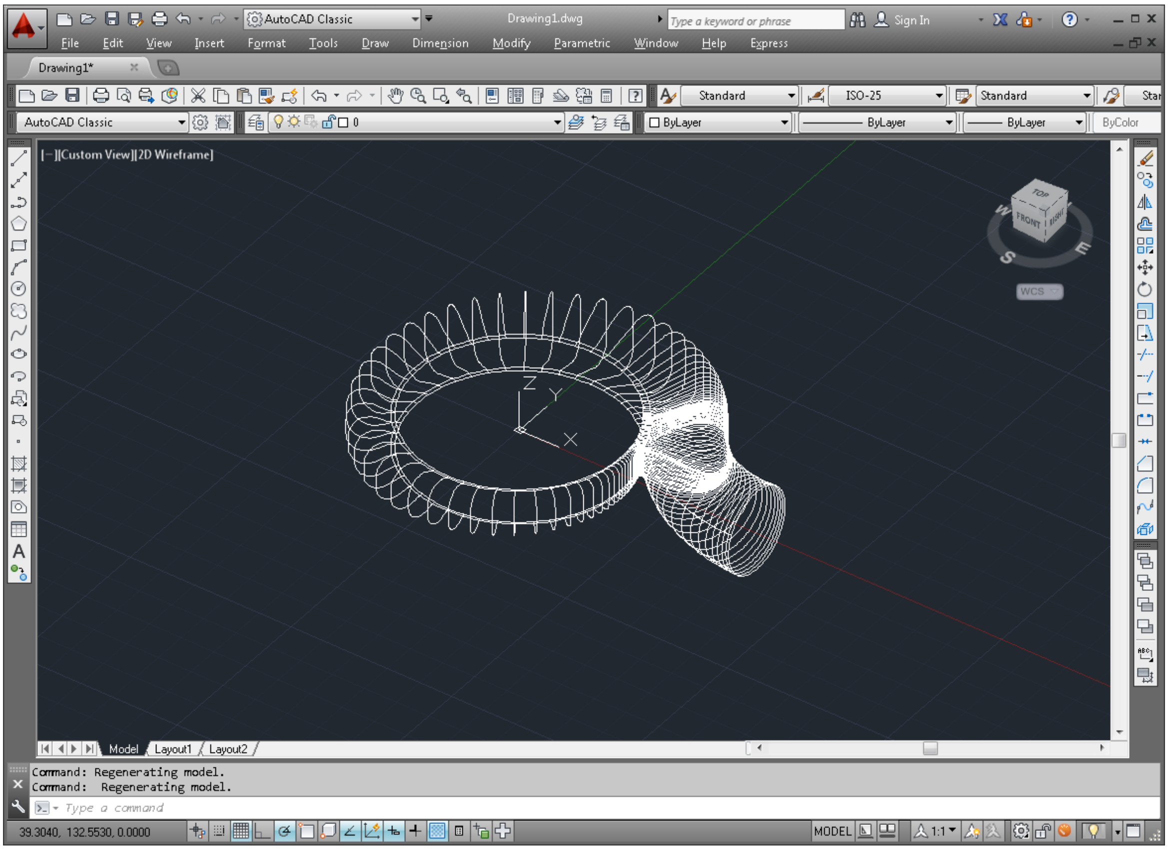
Creating rotational surfaces (Hub, Shroud)

- 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 [STart angle/Reverse/EXpression] <360>:360



Hub and Shroud surfaces

Construction of Volute

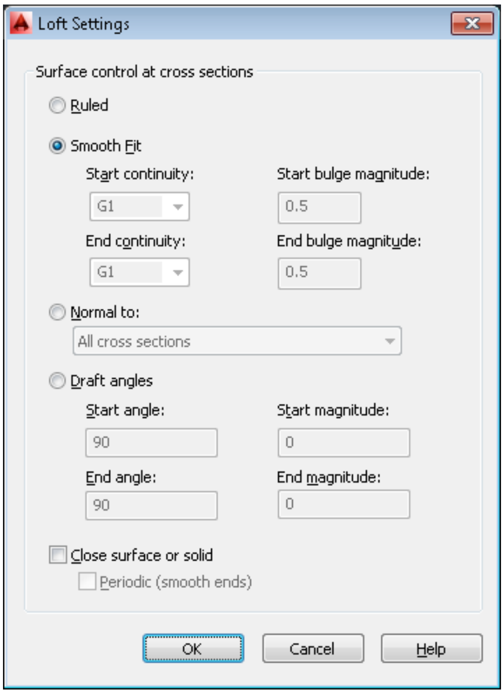

Sample-view after data import

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

6.2.1.4.4.4 CATIA (Dassault Systèmes)

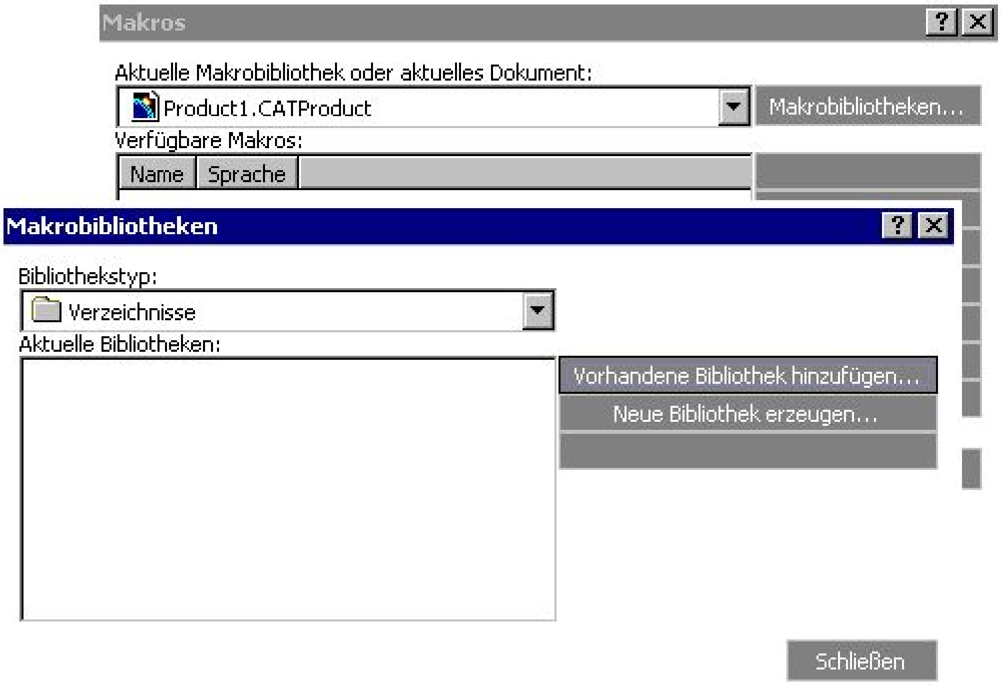
The data-import is realized by a macro that is created for each geometry individually by CFturbo. The macro is loaded and executed in Inventor.

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

6.2.1.4.4.5 Creo Parametric (PTC, Inc.)

The following files are exported by CFturbo for impellers:

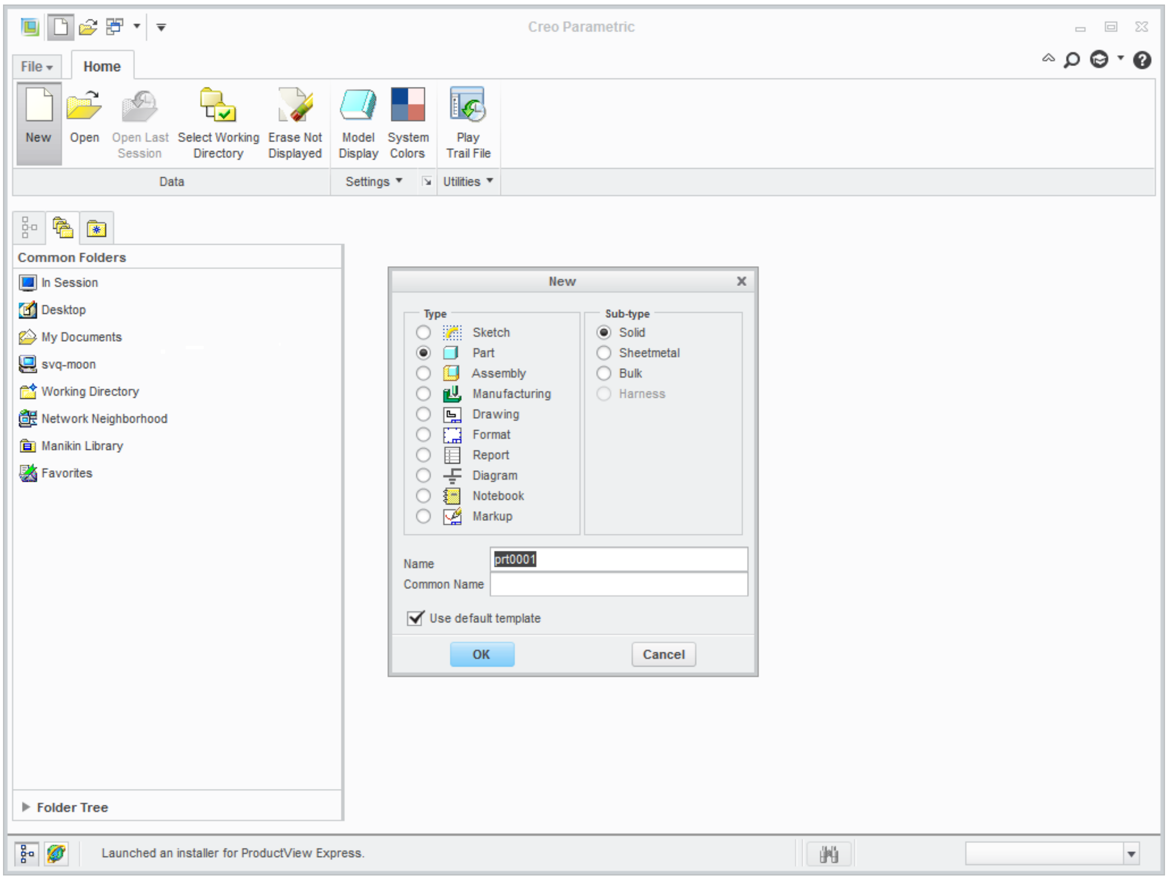
- *-hub.ibl, *-shroud.ibl: points of hub and shroud
- *-profile.ibl: points for blade profiles
- *.ibl: all points for hub, shroud and blades

The following files are exported by CFturbo for volutes:

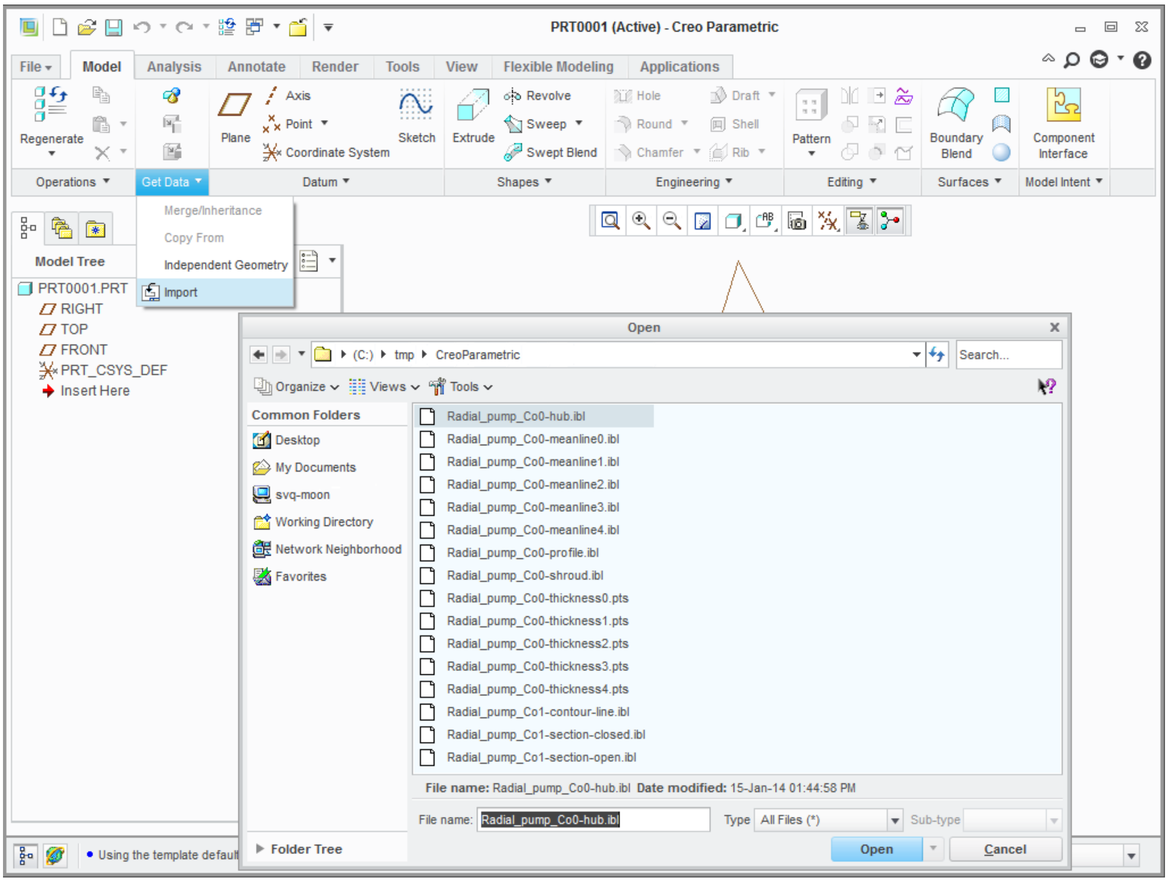
- *-contour-line.ibl: spiral contour points
- *-section-closed.ibl: points for all spiral, cut-water and closed diffuser sections
- *-section-open.ibl: points for all open diffuser sections

Import of curves

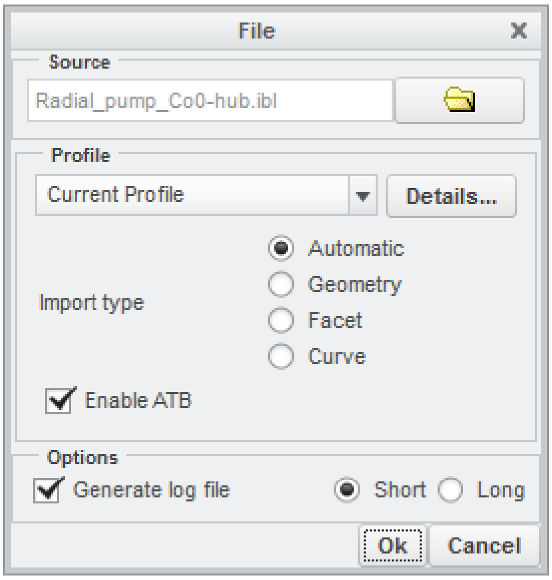
1. Home | New | Part



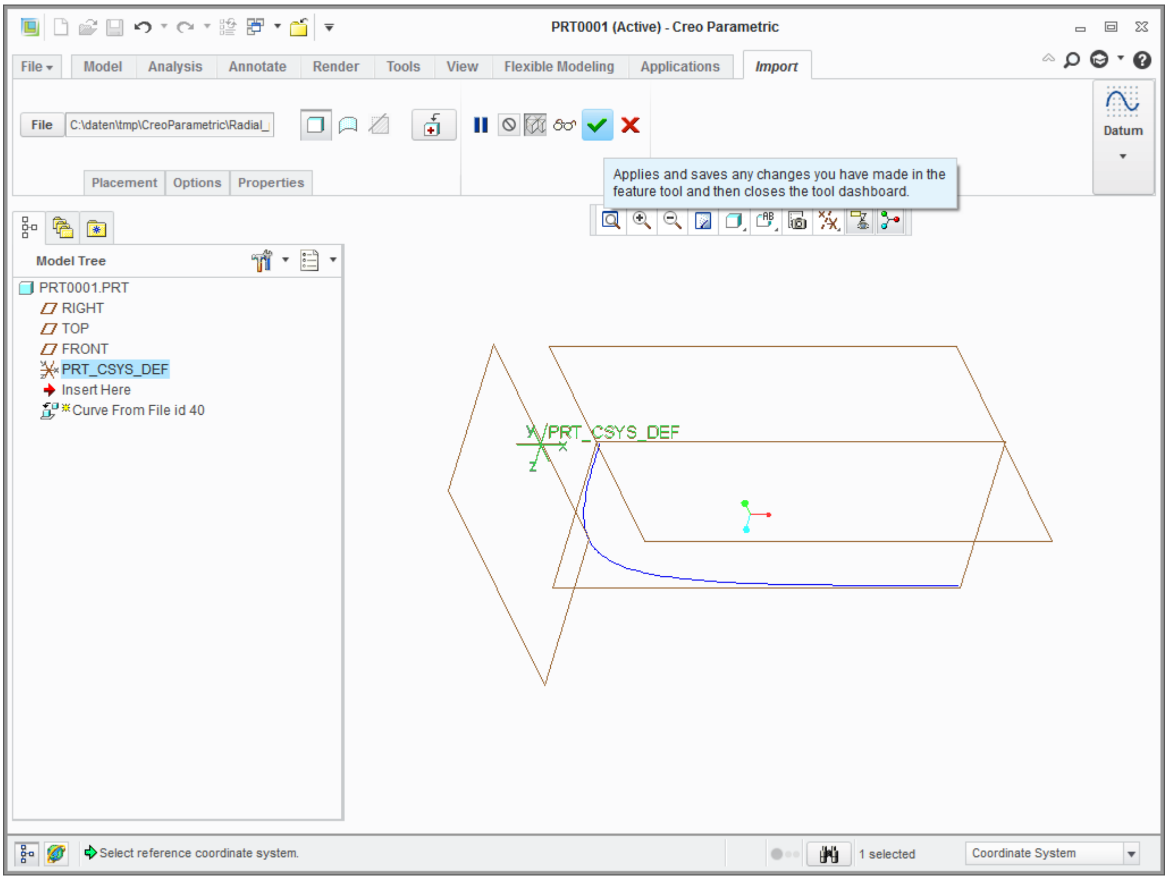
2. Model | Get Data | Import. Select *.pts or *.ibl file



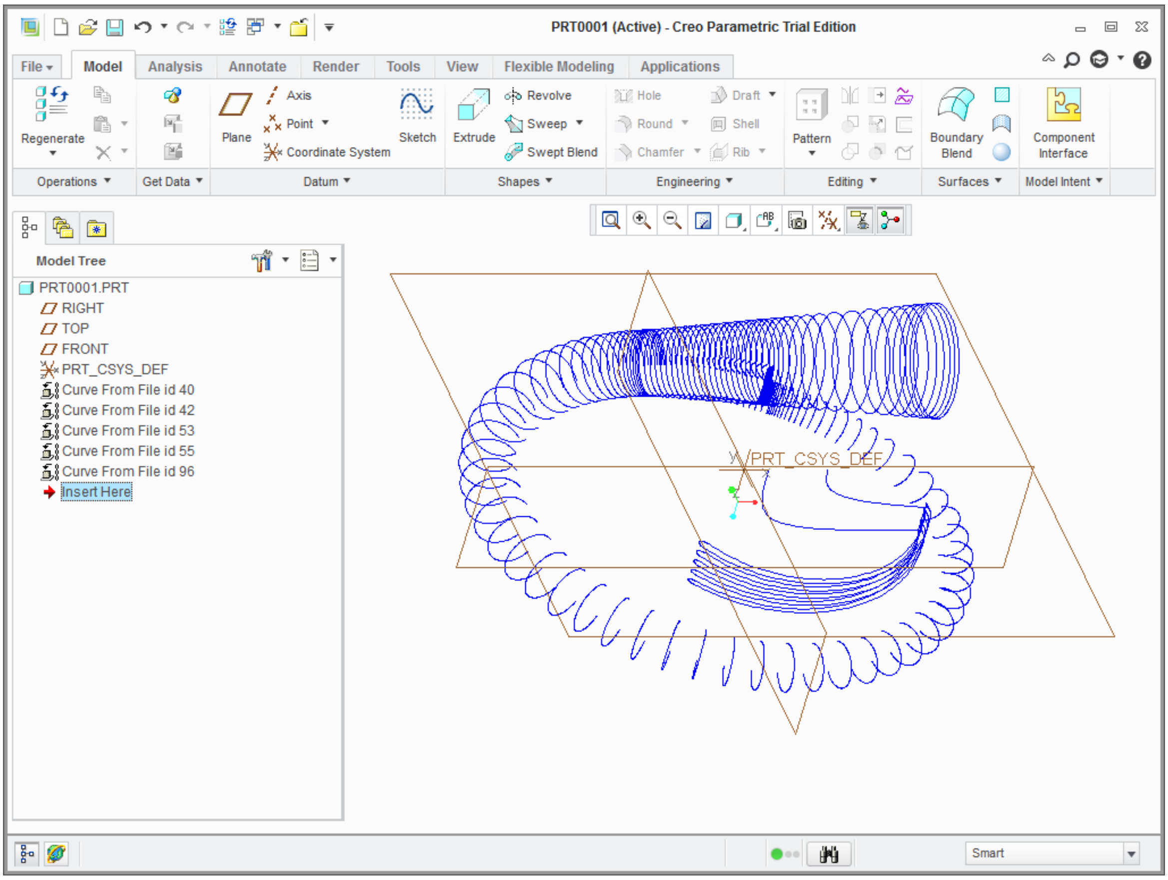
3. In "File" dialog, select desired import options



4. Confirm to finish import process

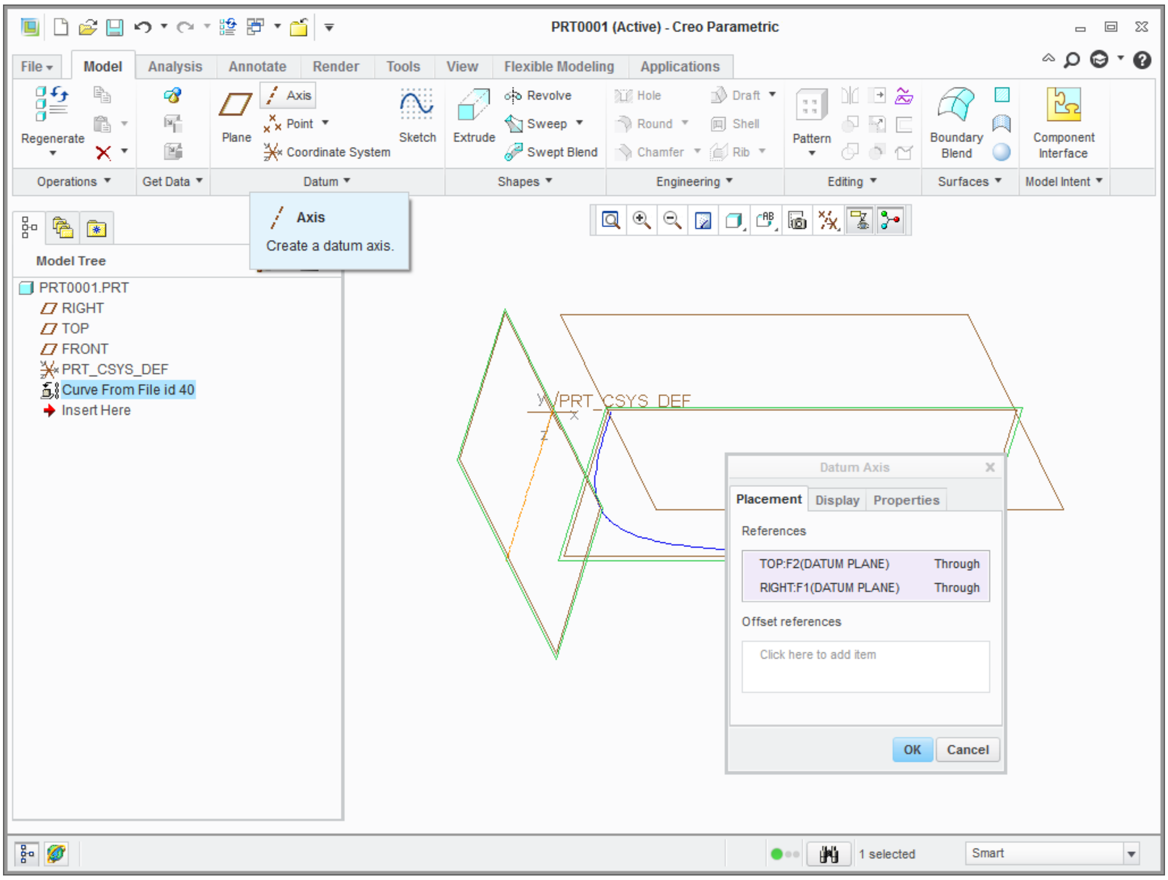


All curves can be imported in this way



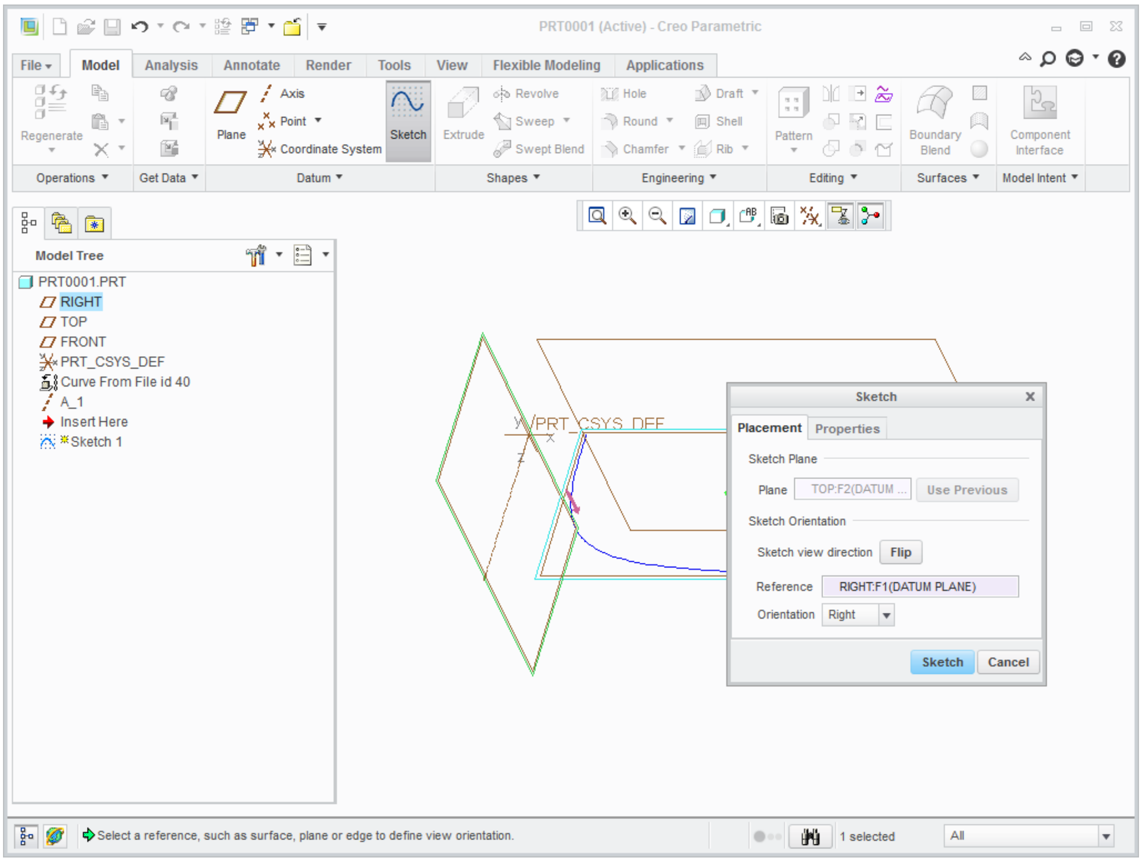
Creating revolution surfaces

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

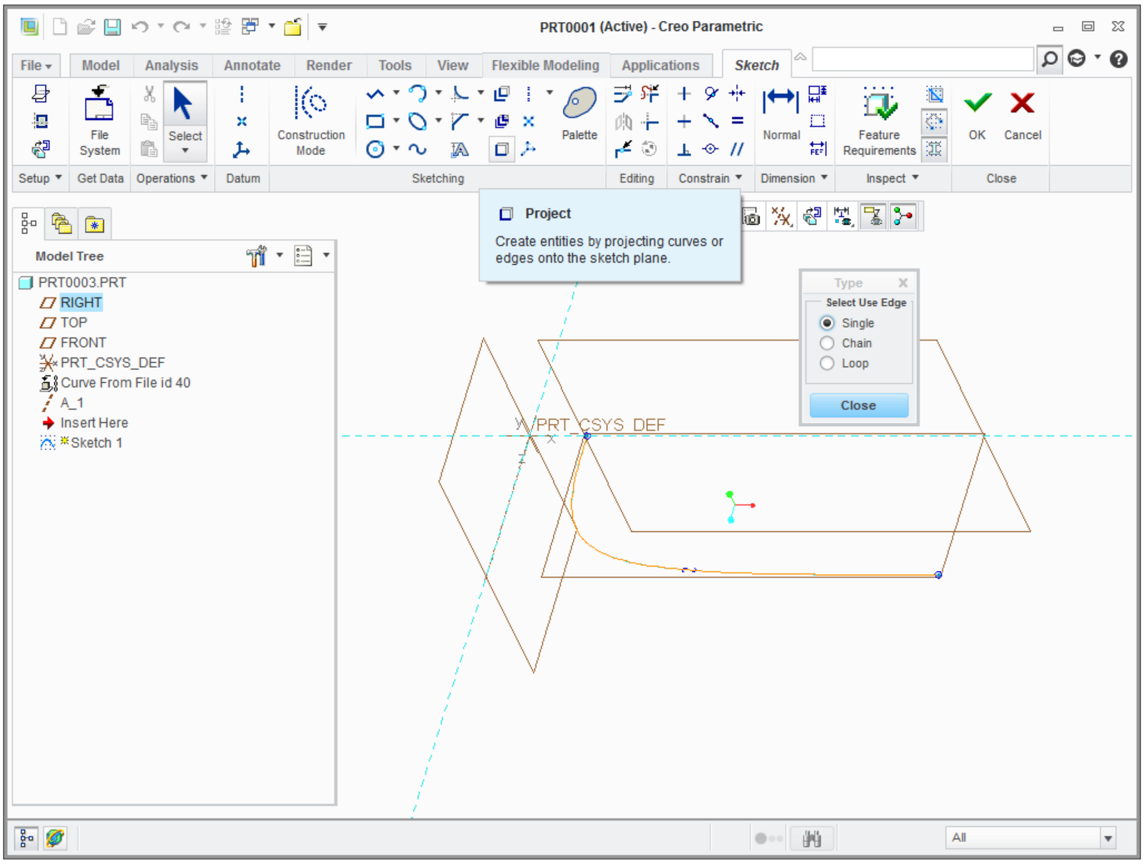


2. Model | Datum | Sketch: create a new sketch

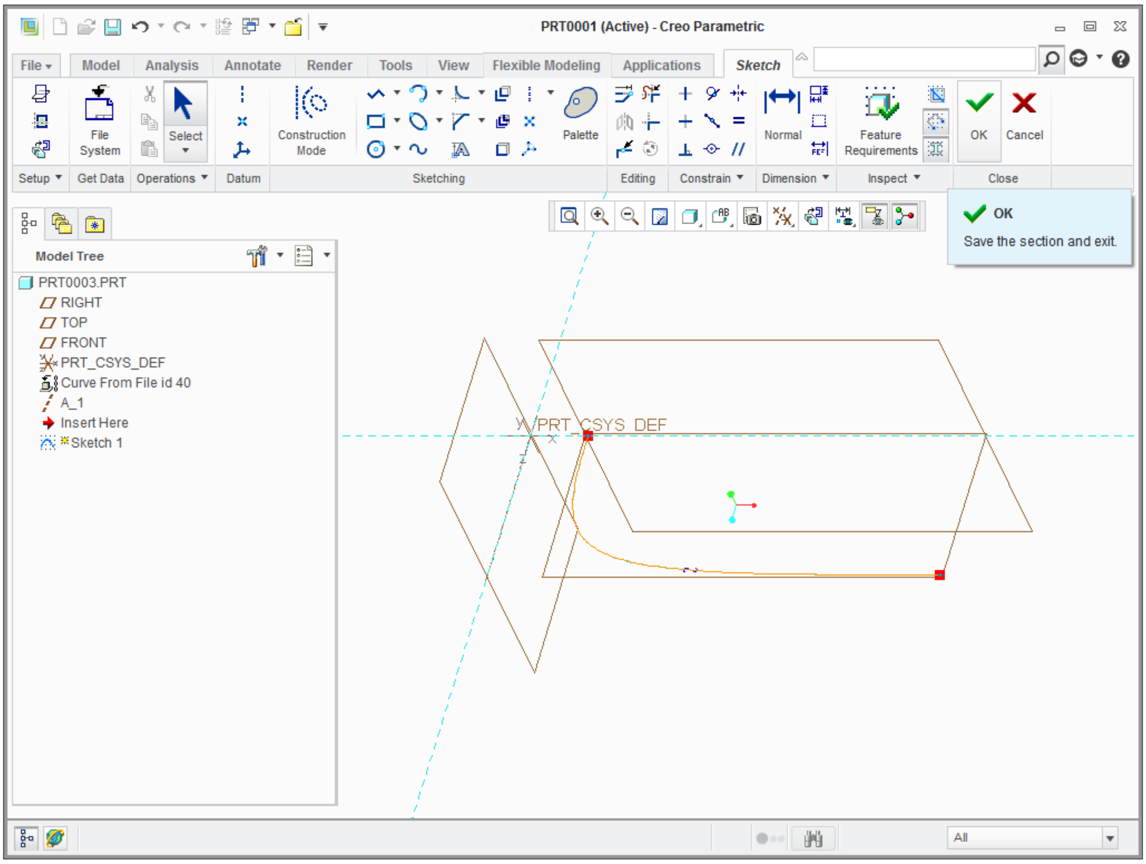
- Select the plane containing the curve to be revolved. Reference and orientation items are set automatically after selection.



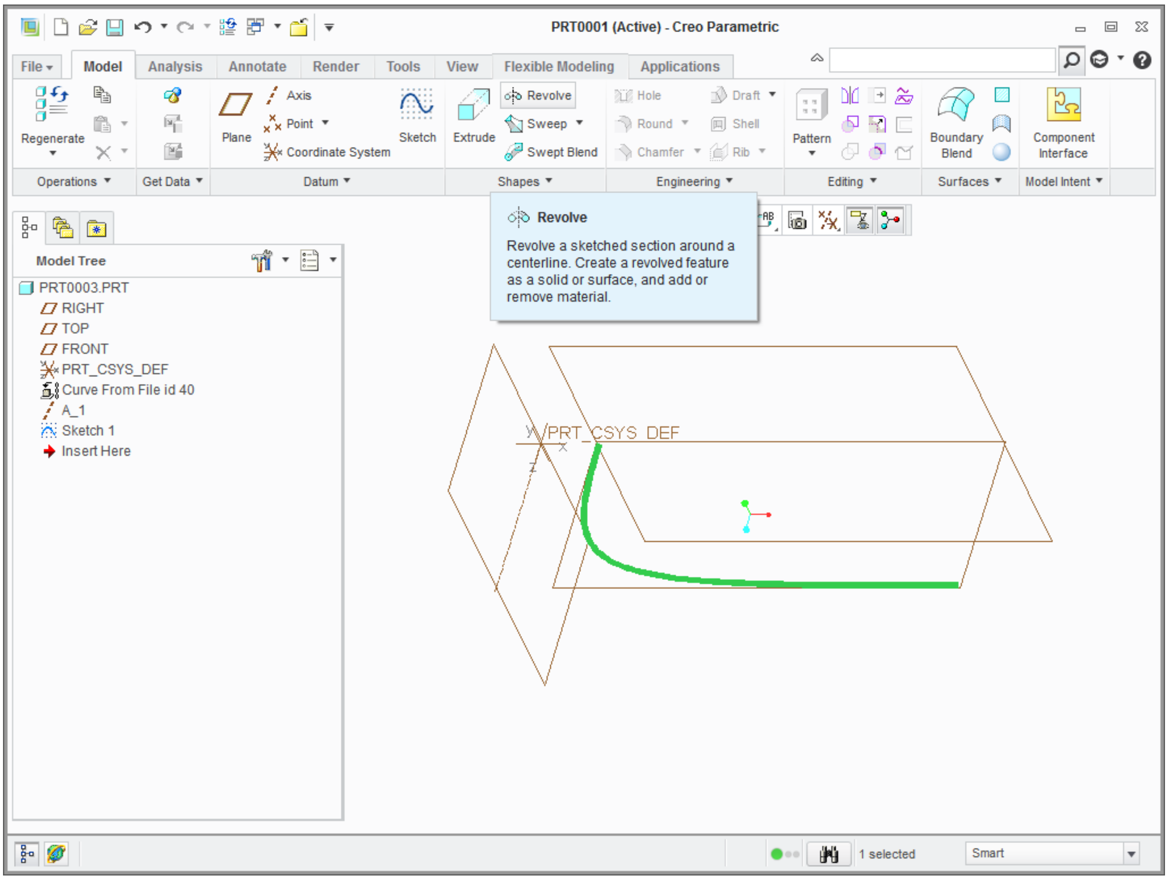
- Sketch | Sketching | Project: do a projection of the curve selecting the curve. Select option "Single" and click on "Close"



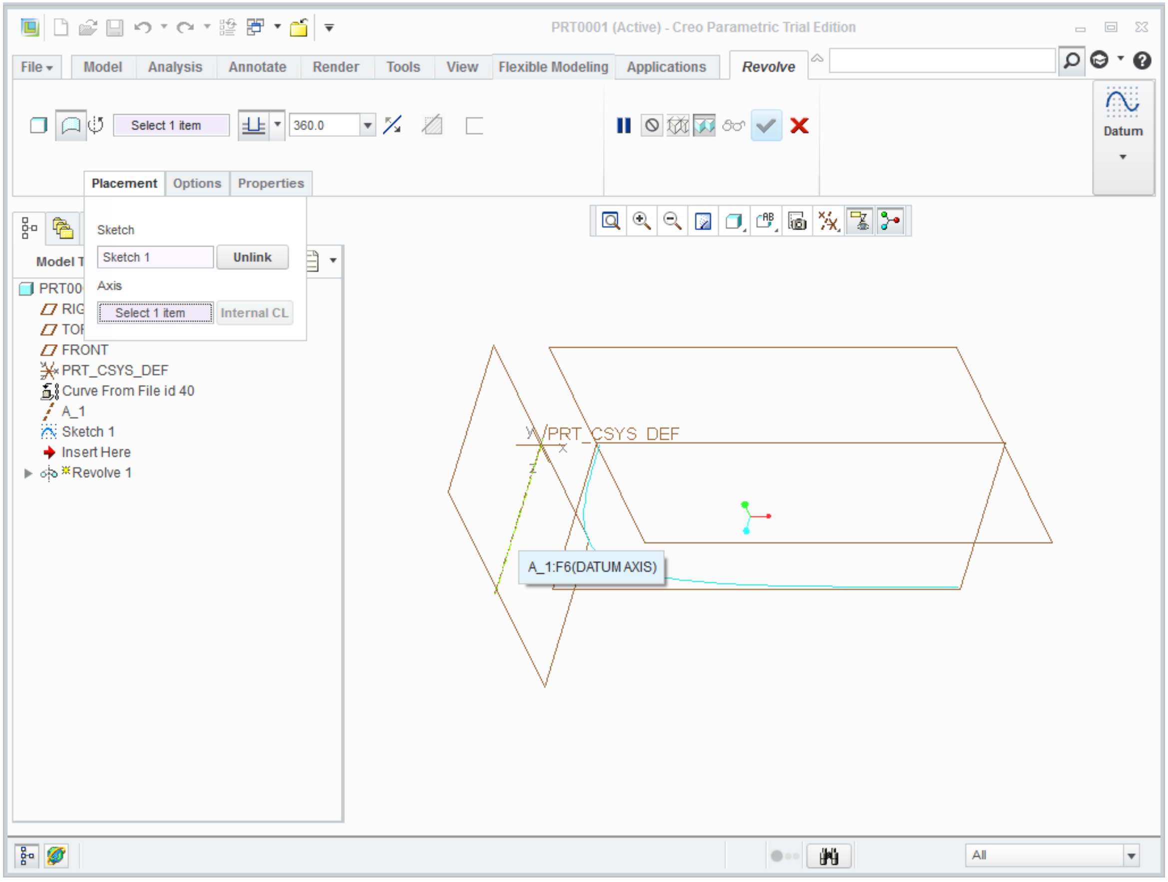
- Finalize sketching task by clicking on "OK"

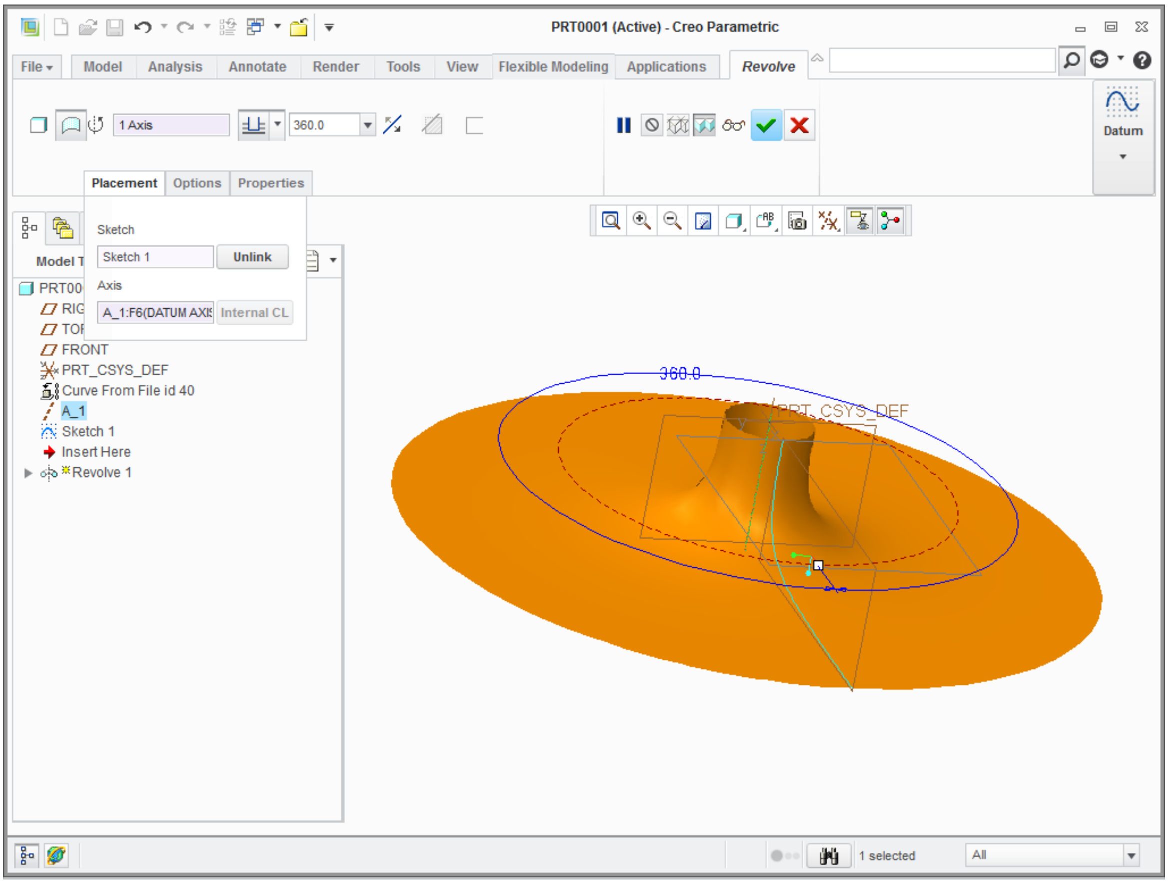


3. Select the curve and click on Model | Shapes | Revolve

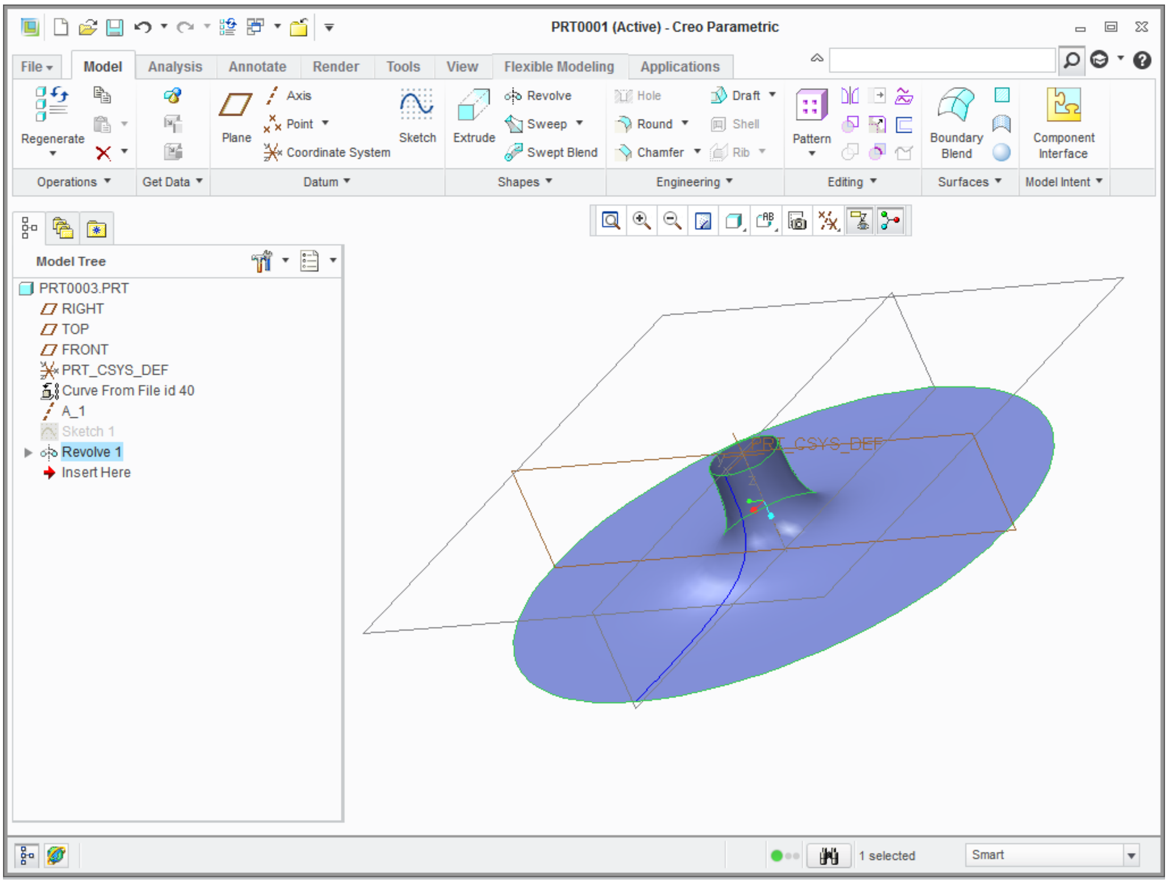


4. Click on field "Axis" under tab "Placements" and select the revolution axis. Surface of revolution will be generated


PRT0001 (Active) - Creo Parametric Trial Edition
File
Model
Analysis
Annotate
Render
Tools
View
Flexible Modeling
Applications
Revolve
Select 1 item
360.0
Datum
Placement
Options
Properties
Sketch
Sketch 1
Unlink
Axis
Select 1 item
Internal CL
PRT00
FRONT
PRT_CSYS_DEF
Curve From File id 40
A_1
Sketch 1
Insert Here
Revolve 1
PRT_CSYS_DEF
A_1:F6(DATUM AXIS)
All



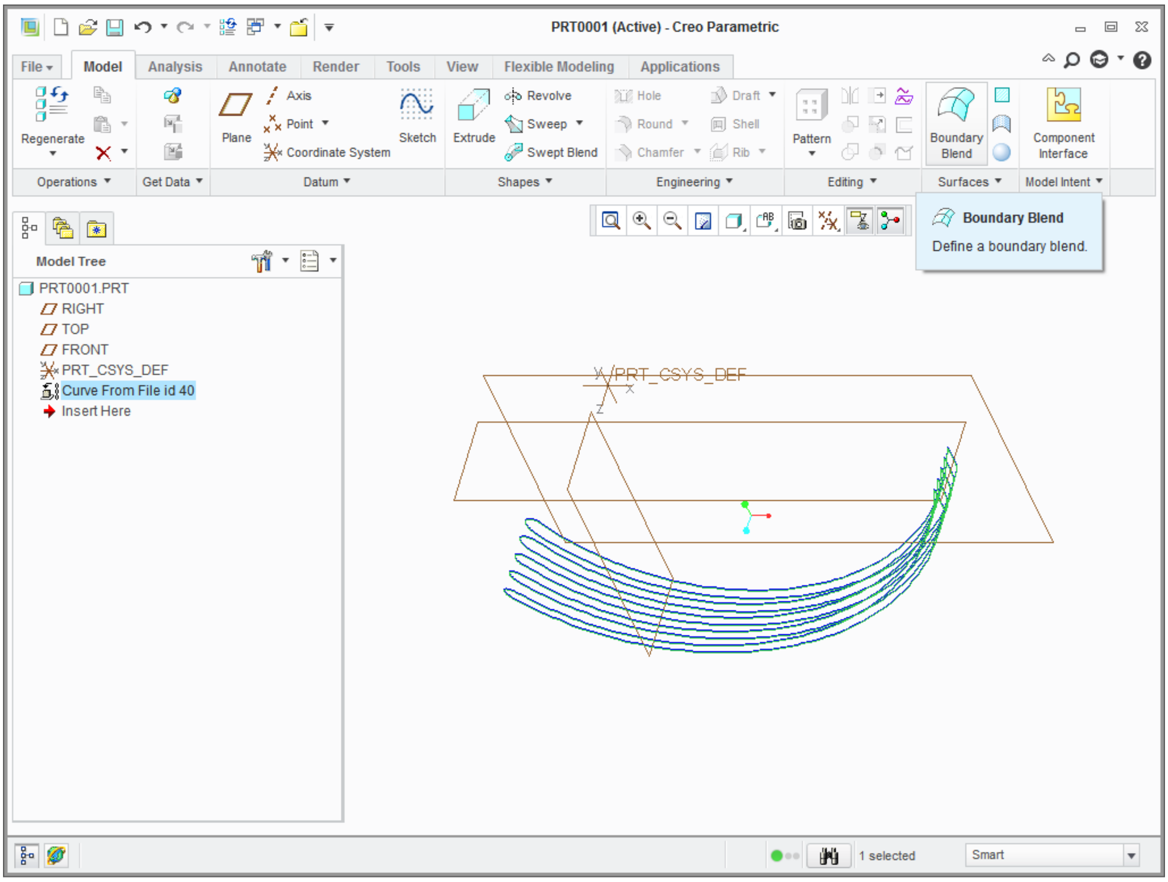
5. Finalize revolve task by clicking on "OK"



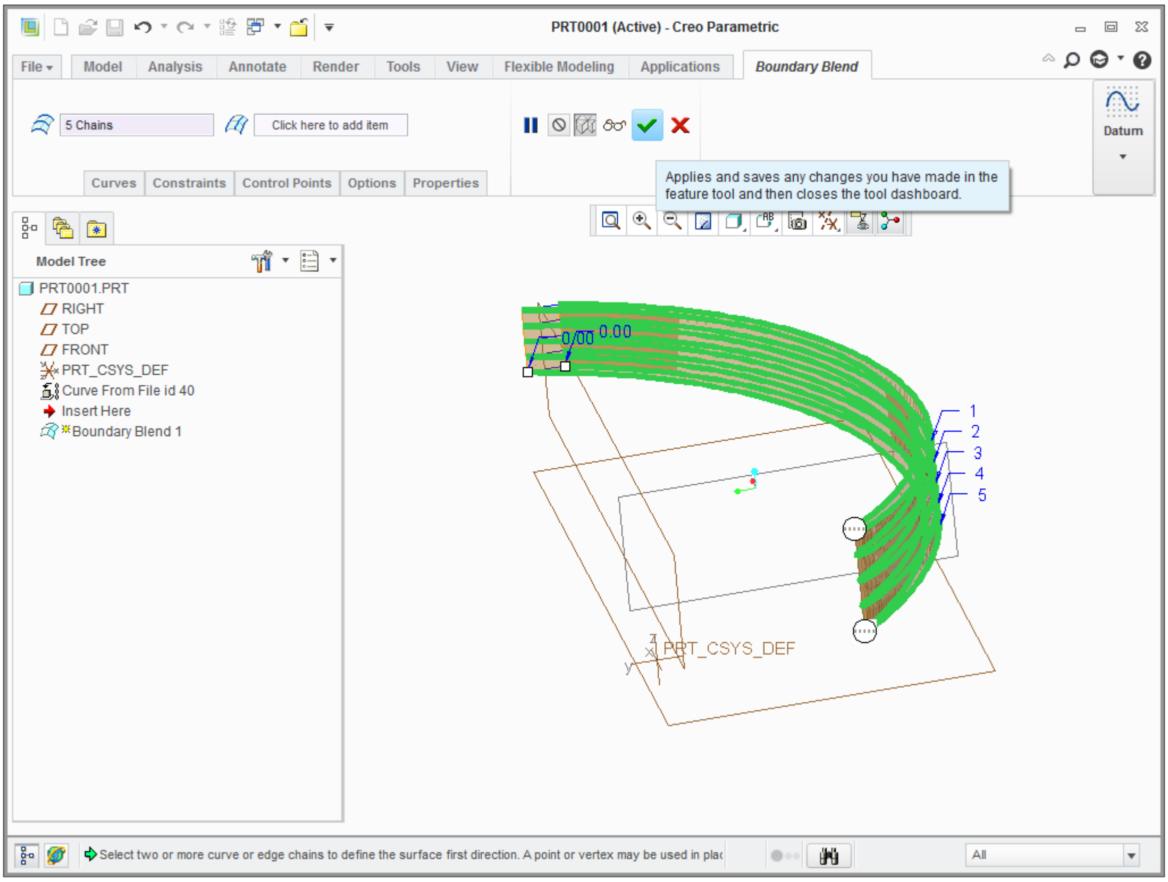
Creating lofted surfaces

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

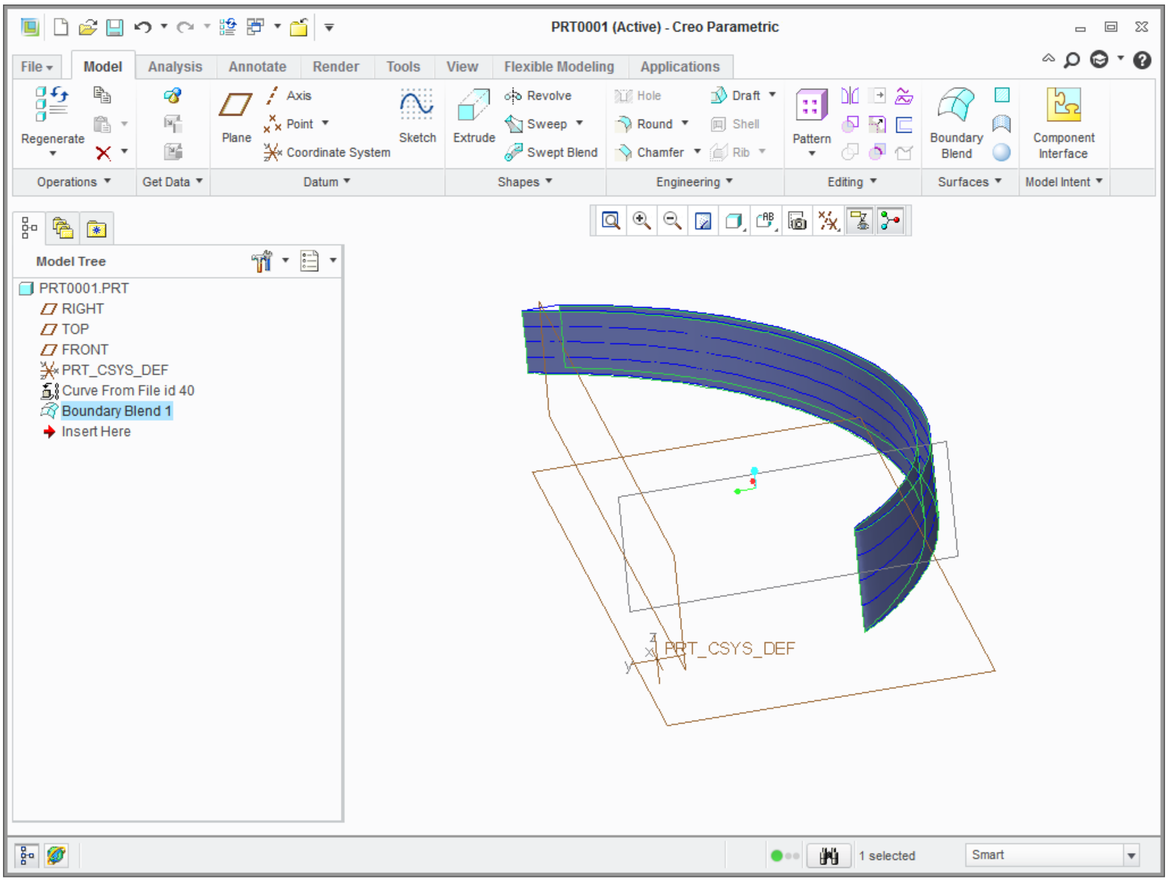
1. Model | Surface | Boundary Blend

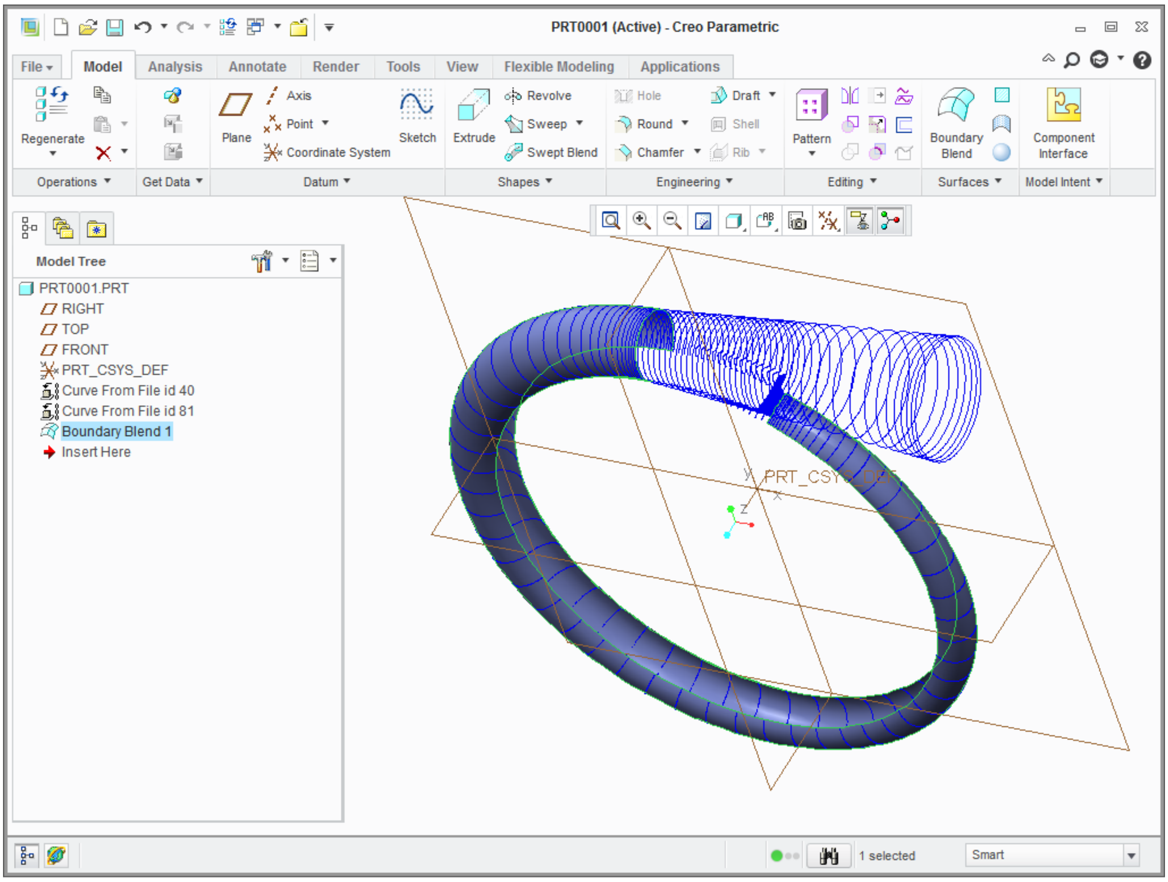


2. Select desired curves (use Ctrl for multi-selection)



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



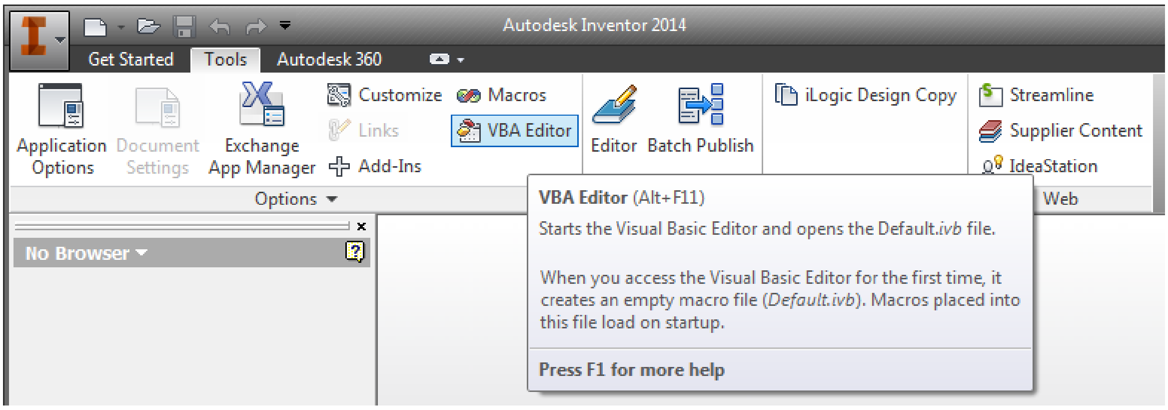


6.2.1.4.4.6 Inventor (Autodesk, Inc.)

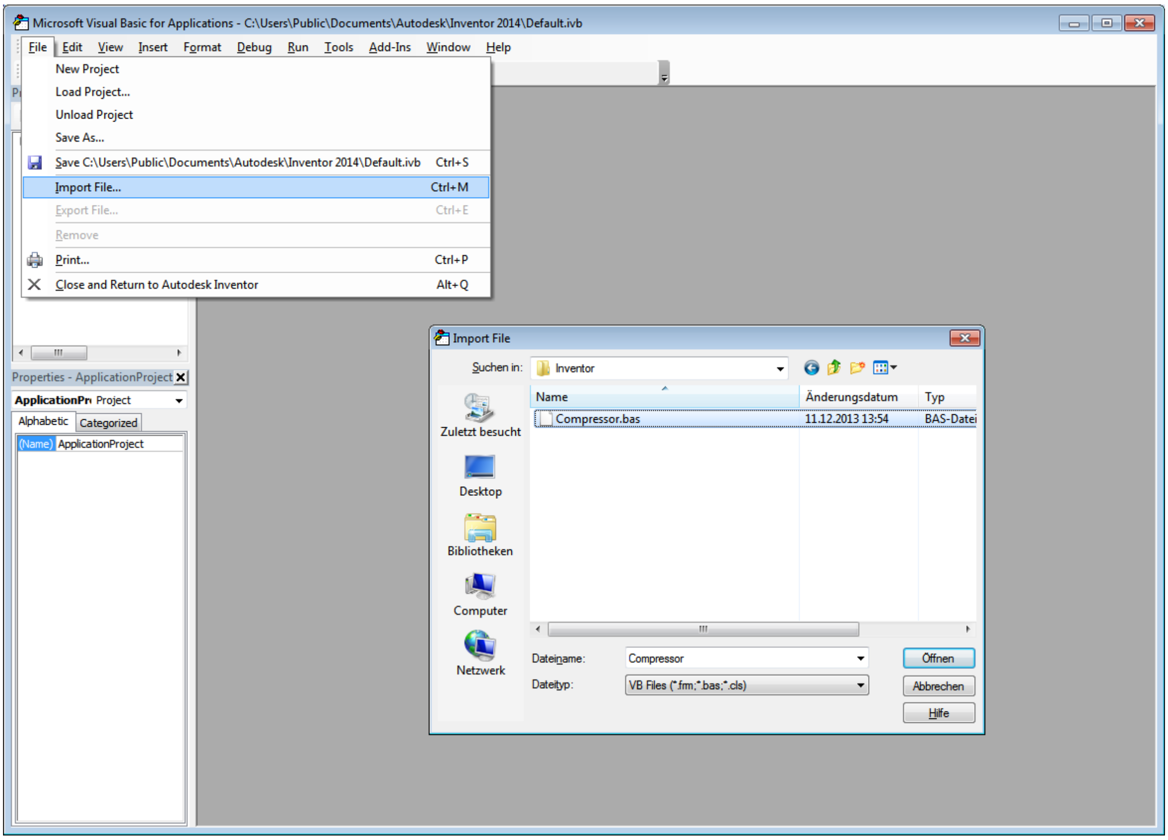
The data-import is realized by a macro that is created for each geometry individually by CFturbo. The macro is loaded and executed in Inventor.

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

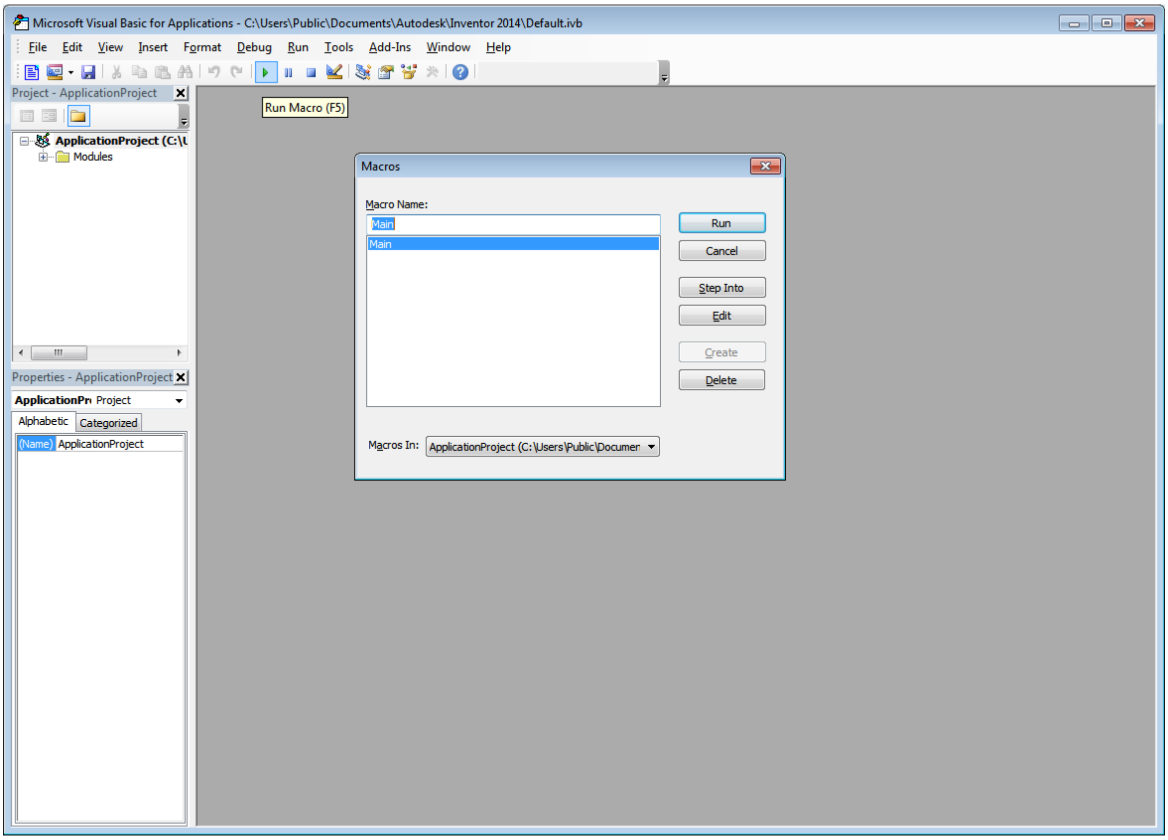
- Tools | VBA Editor



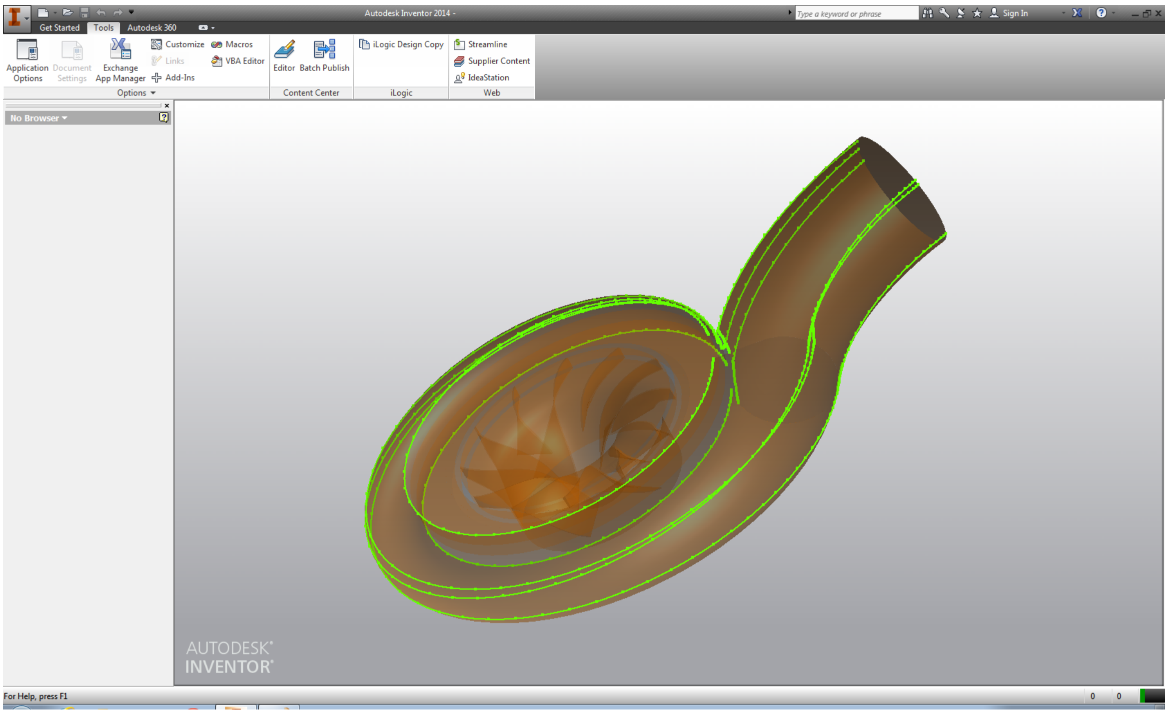
- Open file-open-dialog by *File* | *Import File...* and select *.bas macro-file, possibly a new project has to be created *File* | *New Project*



- Execute imported macro: *Run* | *Run Macro (F5)* close dialog by *Run*



- The time for executing depends on the complexity of the geometry.



Troubleshooting

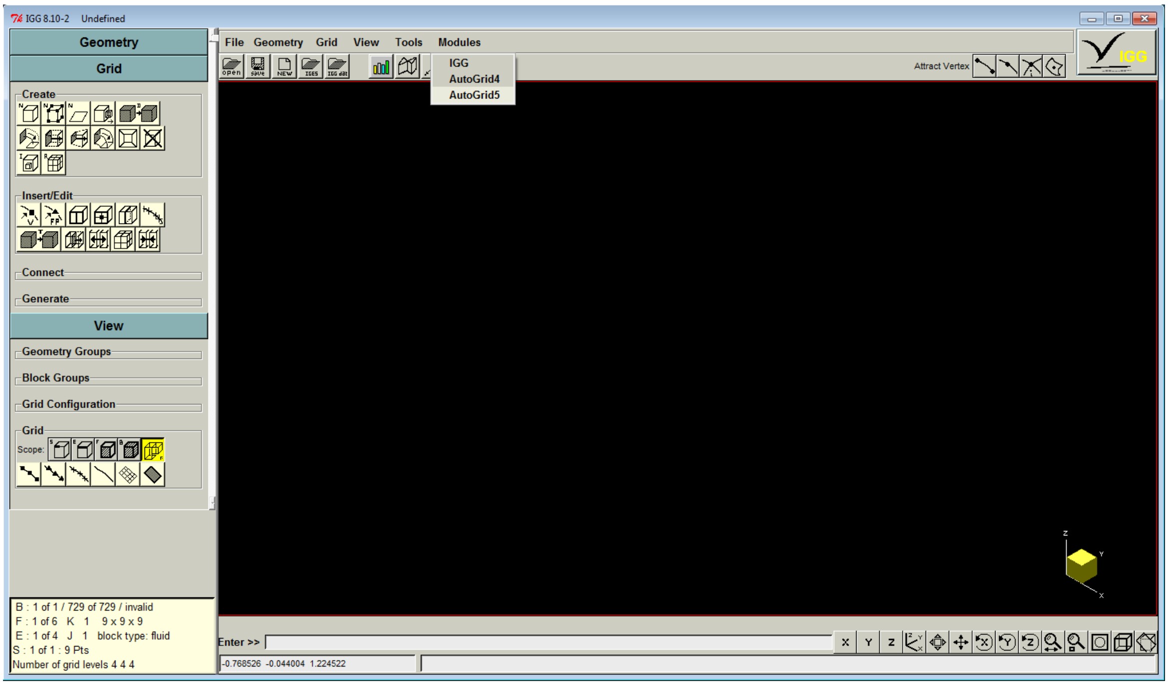
- Selecting the maximal number of points for one or all components in Model settings/Point export 376 could cause too large exported files and "*Out of memory*" error message while importing in Inventor:

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

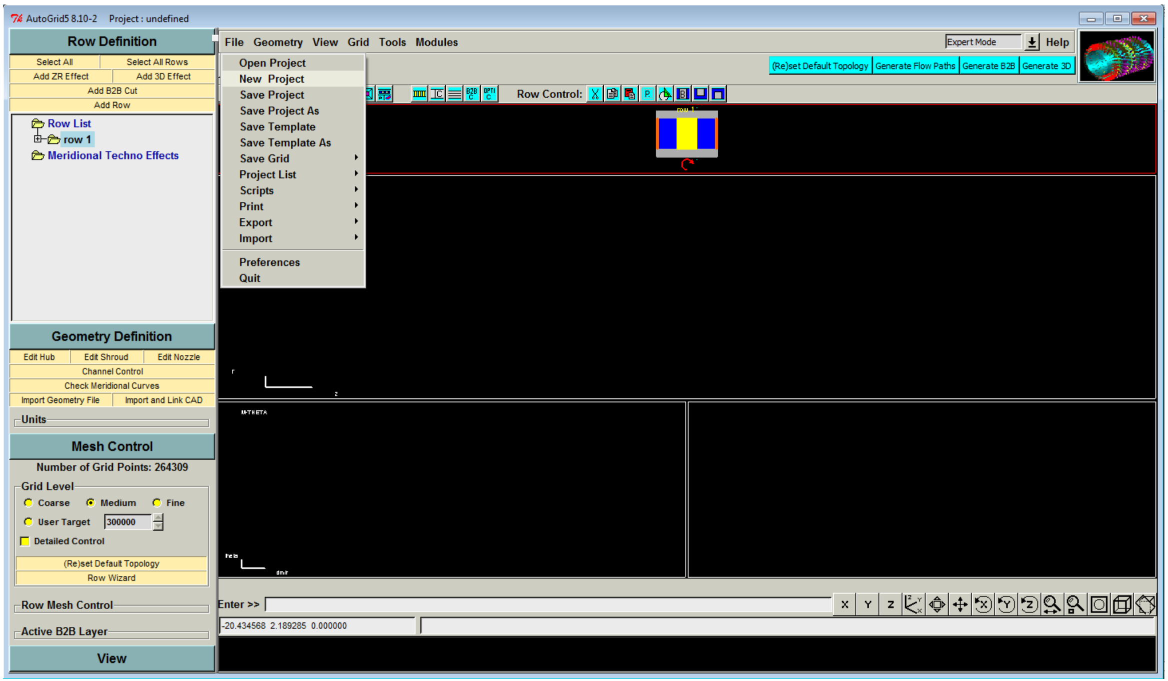
6.2.1.4.4.7 AutoGrid (NUMECA International)

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

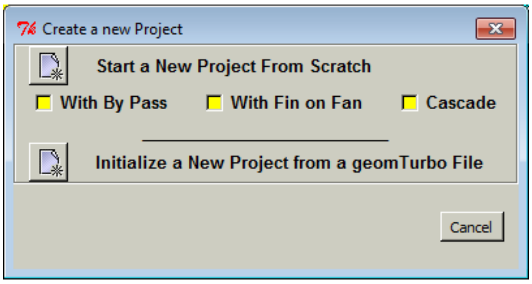
- Start IGG
- Change to AutoGrid5-mode: *Modules* | *AutoGrid5*



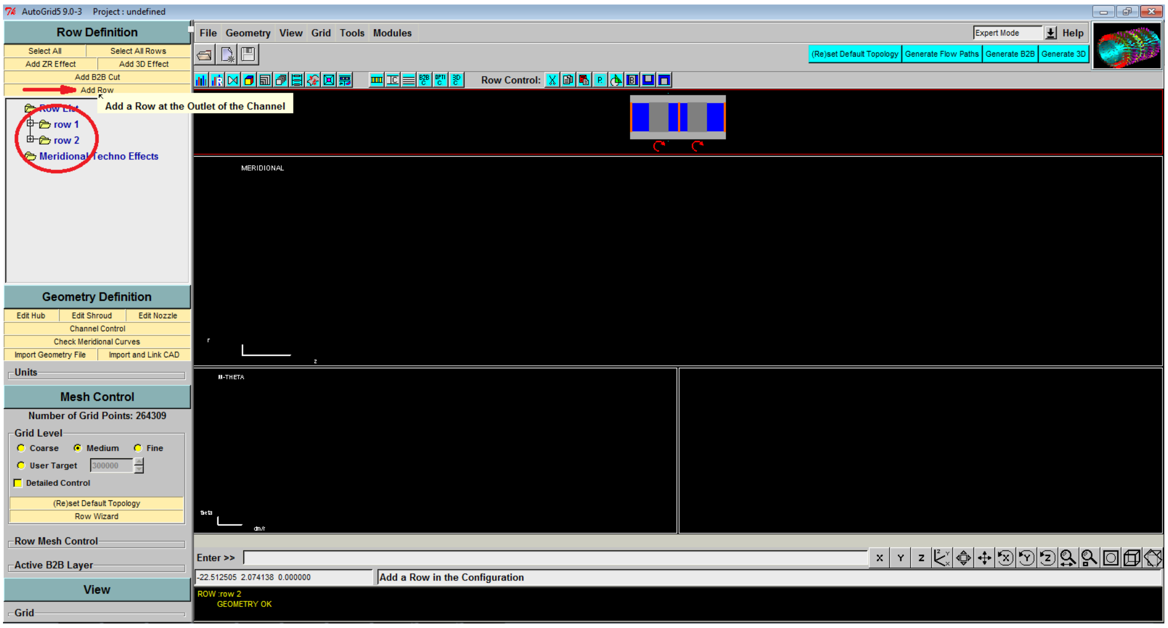
- Open a new project: *File* | *New Project*



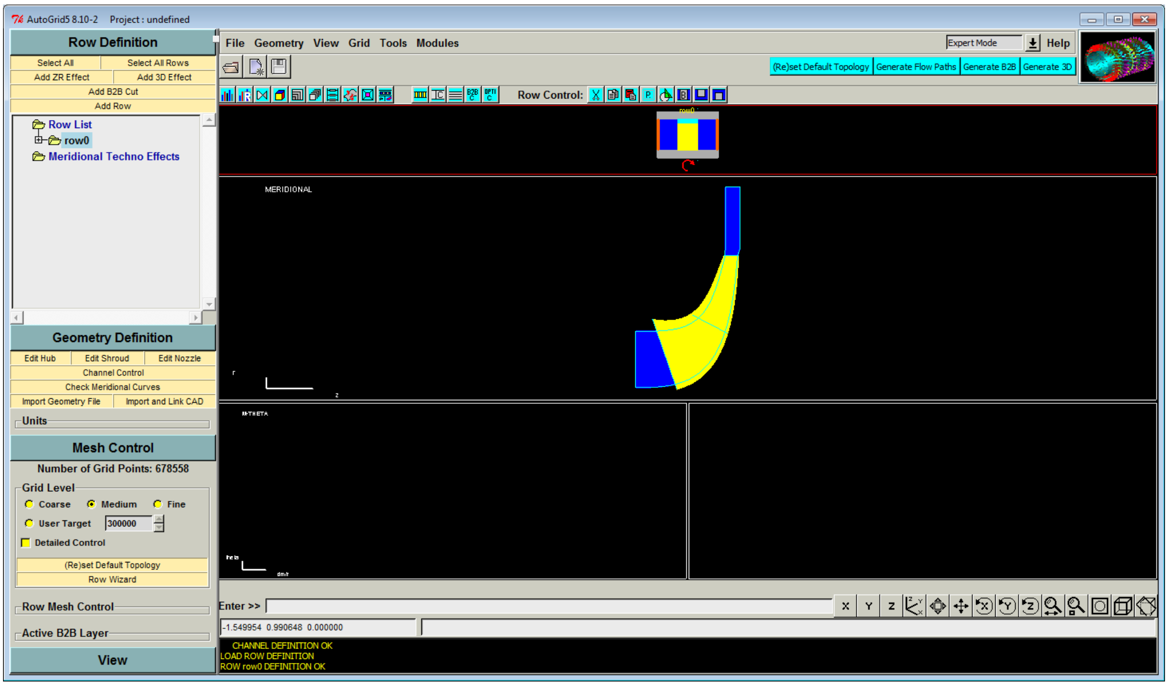
- Close dialog by *Initialize a New Project from a geomTurbo File*



- 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.

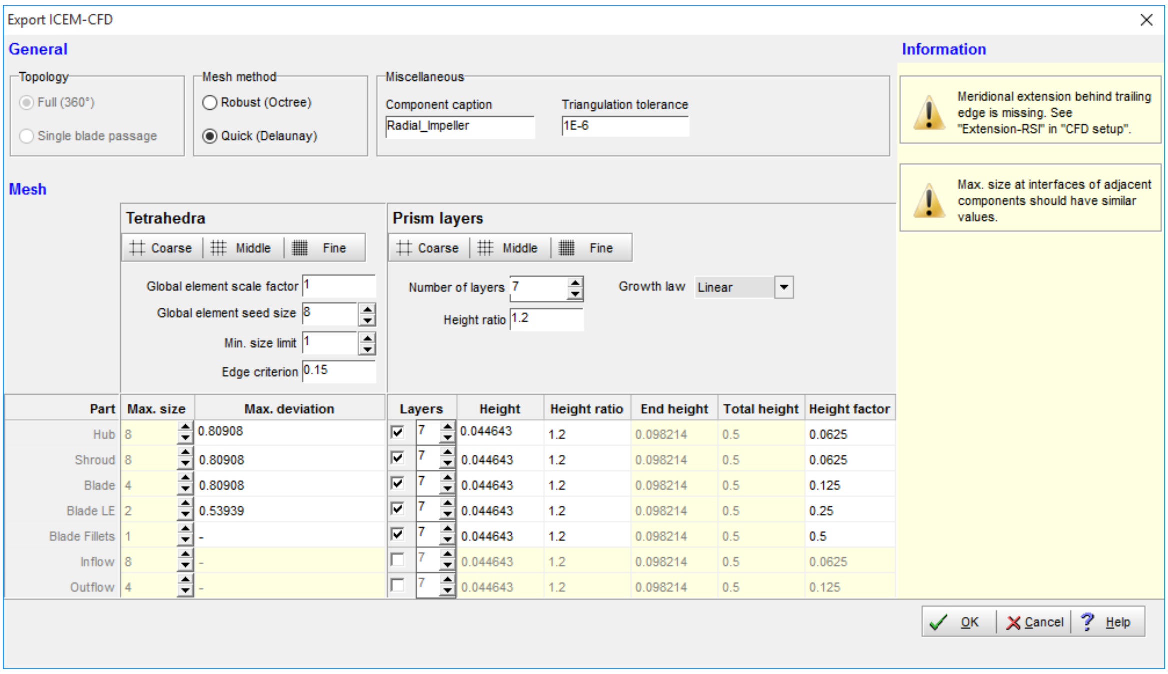
6.2.1.4.4.8 ICEM CFD (ANSYS)

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

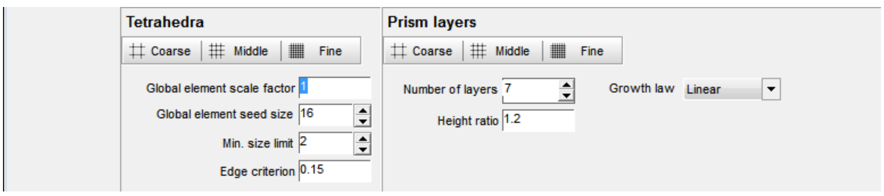
CFturbo2ICEM is a script solution for automatic geometry generation and meshing of CFturbo components.

Export to ICEM-CFD is used for the CFturbo2ICEM scripts only.

2 files are exported: a *.tinXML flie containing all meshing parameters specified in CFturbo and a *.stp file containing the designed geometry with specific naming conventions. Detailed description of the parameters can be found on the available documentation.



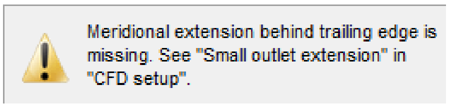
Global settings



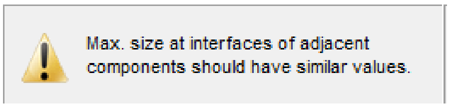
Local settings

Part	Max. size	Max. deviation	Layers	Height	Height ratio	End height	Total height	Height factor
Hub	16	0.3971	☑ 7	0.08929	1.2	0.19643	1	0.0625
Shroud	16	0.3971	☑ 7	0.08929	1.2	0.19643	1	0.0625
Blade	8	0.3971	☑ 7	0.08929	1.2	0.19643	1	0.125
Blade LE	4	0.26471	☑ 7	0.08929	1.2	0.19643	1	0.25
Blade TE	2	0.26471	☑ 7	0.08929	1.2	0.19643	1	0.5
Inflow	16	-	☑ 7	0.08929	1.2	0.19643	1	0.0625
Outflow	8	-	☑ 7	0.08929	1.2	0.19643	1	0.125

Possible warnings:



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

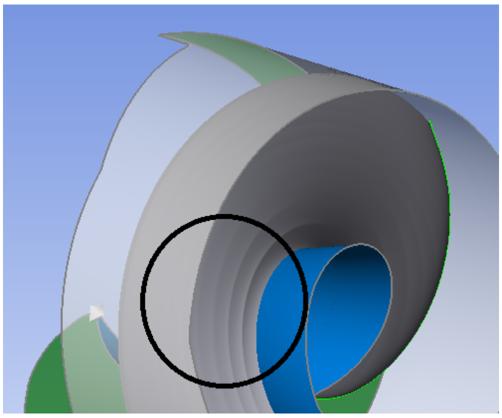


Cell size at the interface between neighboring components should be similar.

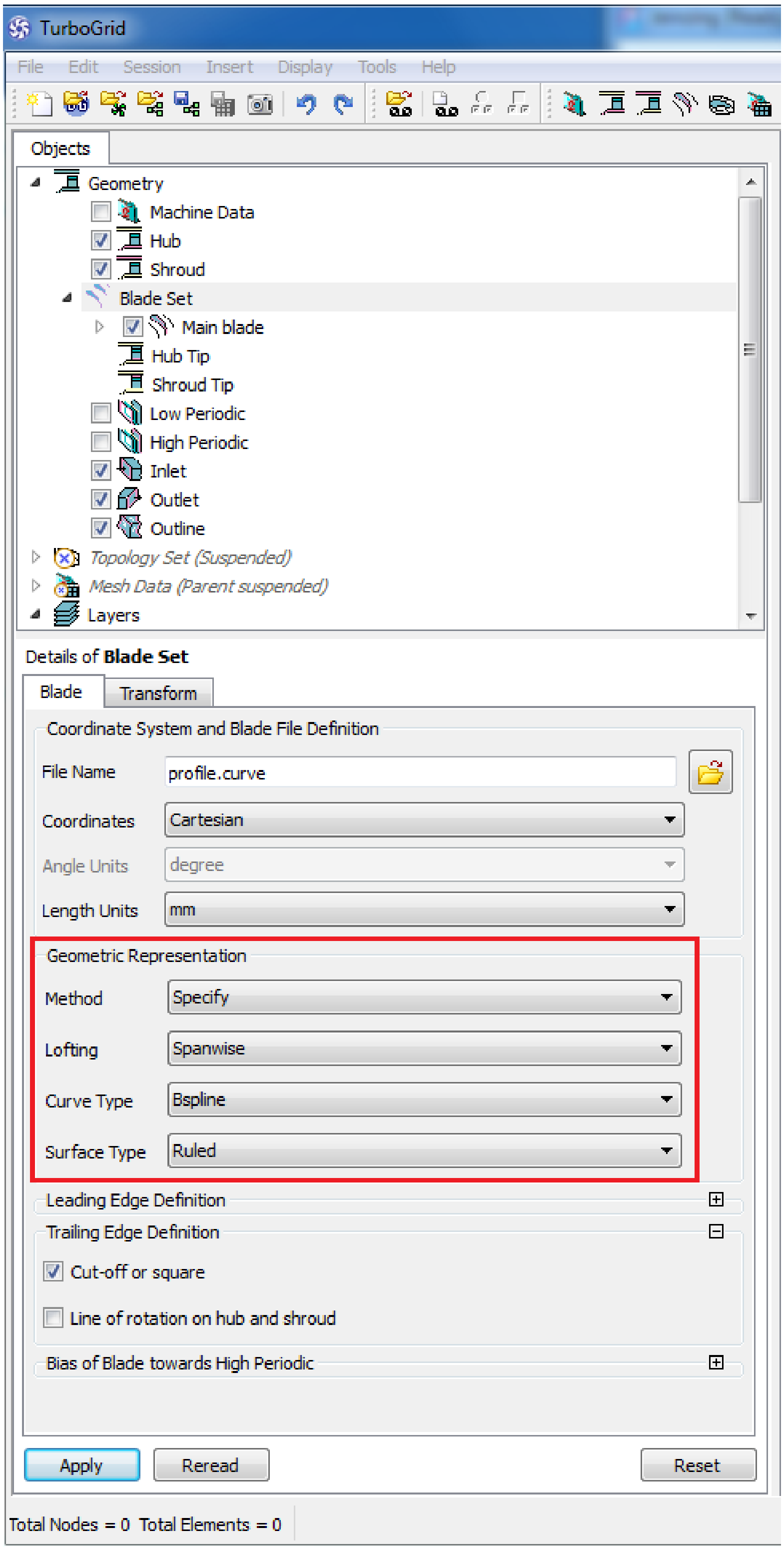
6.2.1.4.4.9 TurboGrid (ANSYS)

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.*"

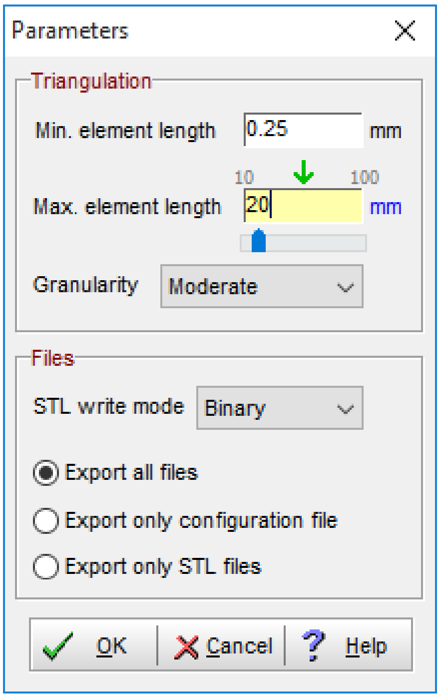
Two options are available to increase this region:

a) moving the leading/ trailing edge in meridional contour. The edge has not to be fixed on inlet/ outlet 270. 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 368. This option does not incur a geometrical modification of the component but of the neighboring one if exists.

6.2.1.4.4.10 Simerics

In addition to the STL-Parameters 100, three export options are available:



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

Export only configuration file: STL files are not exported. This option is useful if only the configuration file is desired because the STL files are already available. This saves time because the geometry does not have to be triangulated.

Export only STL files: 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.

6.2.1.4.5 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 is then 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

(1) in the CFturbo installation directory: in the directory **Examples**

(2) on the CFturbo website: http://www.cfturbo.com/download.html

6.2.1.5 Import 3D geometry

? Project | Project | Import 3D

The 3D Import enables the user to view 3D data in IGES, STEP, STL 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 172 and can be transformed 180 and exported.

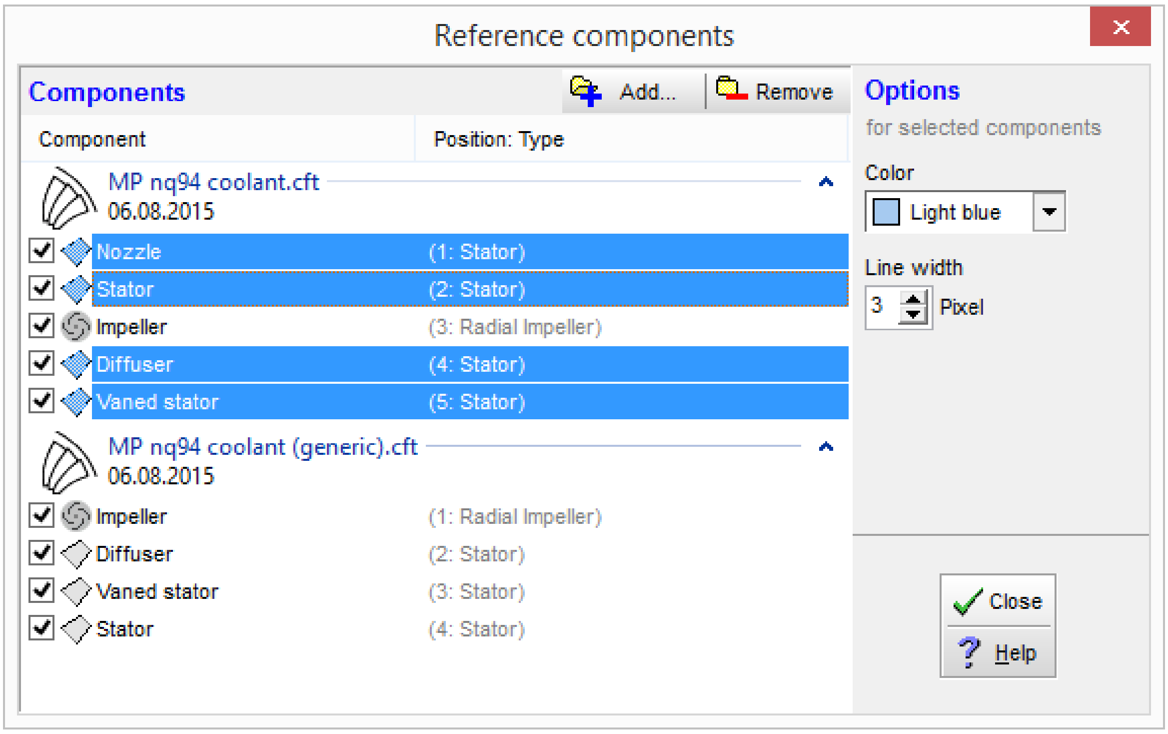
Imported CFturbo-projects are a pure 3D data import. The structure of geometrical parts is visible in the 3D model tree 179, but no design steps can be modified.

If the the import consumes a lot of time, a lower resolution can be selected (see also Model display 173).

6.2.1.6 Reference components

? Project | Project | Reference components

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



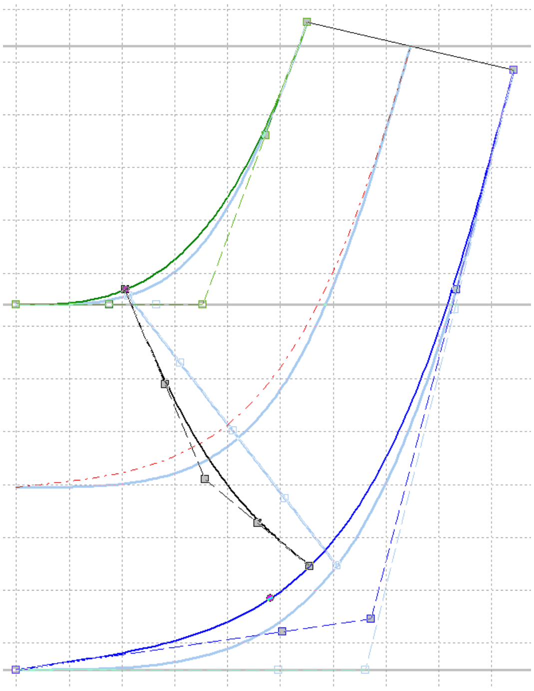
Using the **Add**-button any reference project (*.CFT- file) can be added. All components of the reference project are grouped under the selected file name.

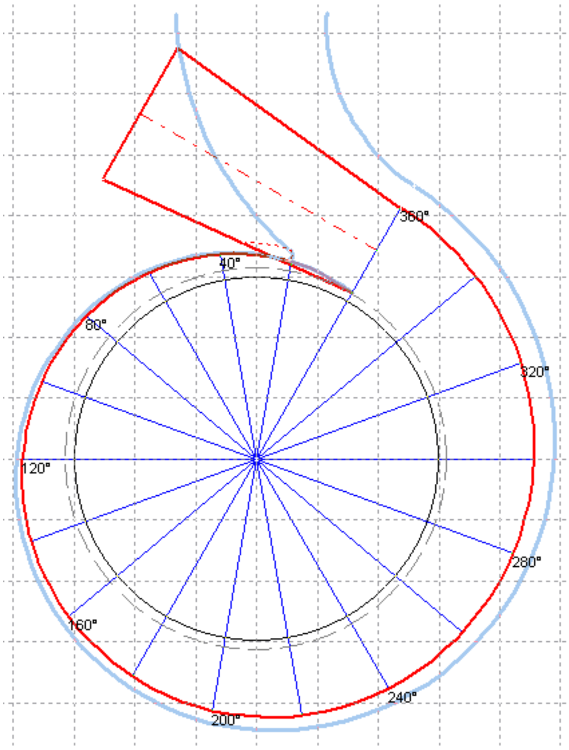
Each component has its own color and line width (panel **Options**). Multiple components can be selected using <Shift> and <Ctrl> keys. Clicking on the group header area selects all components of the corresponding project, <Ctrl> <A> selects all components.

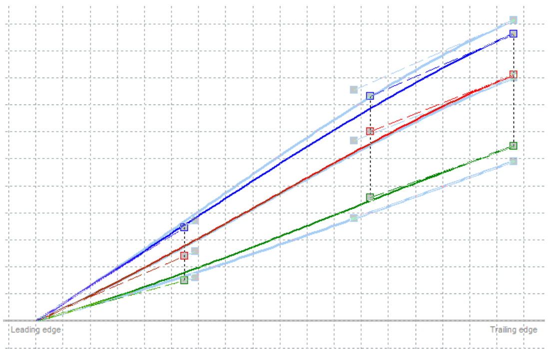
With the **Remove**-button the selected reference project with all its components can be deleted from the list. However single reference components may be deactivated by the check box at the beginning of the line.

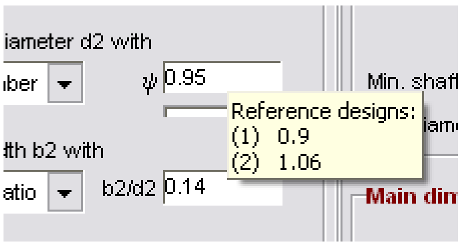
Display in dialogs

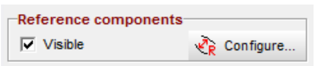
Reference geometries are displayed in the dialogs with selected color and line width. Numerical values appear as small hints on input fields when mouse is moved over it.











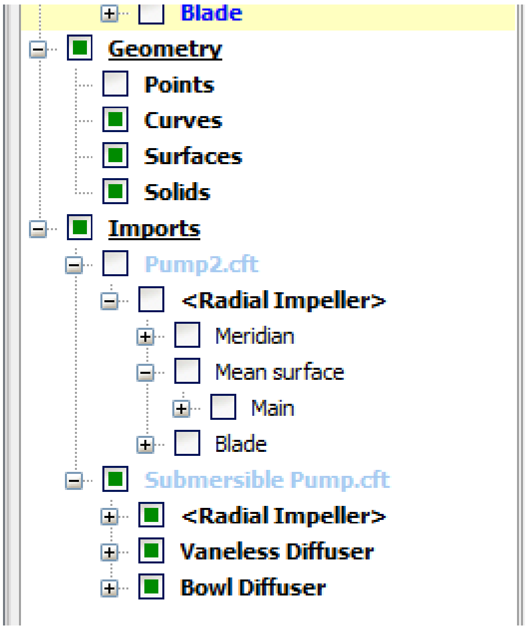
Down right in the design step dialog windows you could completely switch off the display of reference geometries and start the configuration dialog.

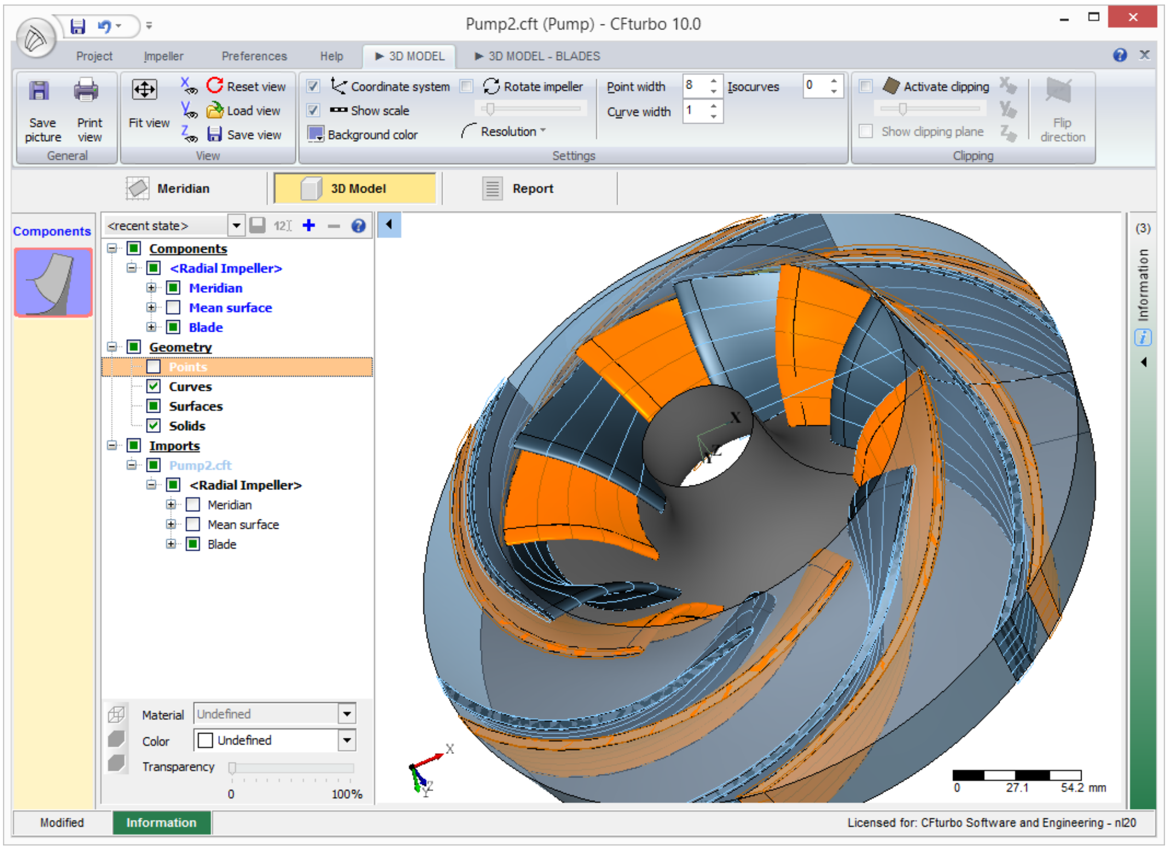
Please note: If you add reference designs in a design step dialog the imported geometry could be invisible initially if it's far away from the currently designed geometry. There is no automatic scaling of the diagram.

Display in 3D-model

Reference geometry is displayed as 3D model additionally.

All reference geometries are arranged in the model tree in the region "Imports", whereas the single parts can be configured like the normal geometry.





6.2.1.7 Show/Hide messages

? Project | Project | Show/ Hide messages

This button shows or hides the message panel on the right side of the main window.

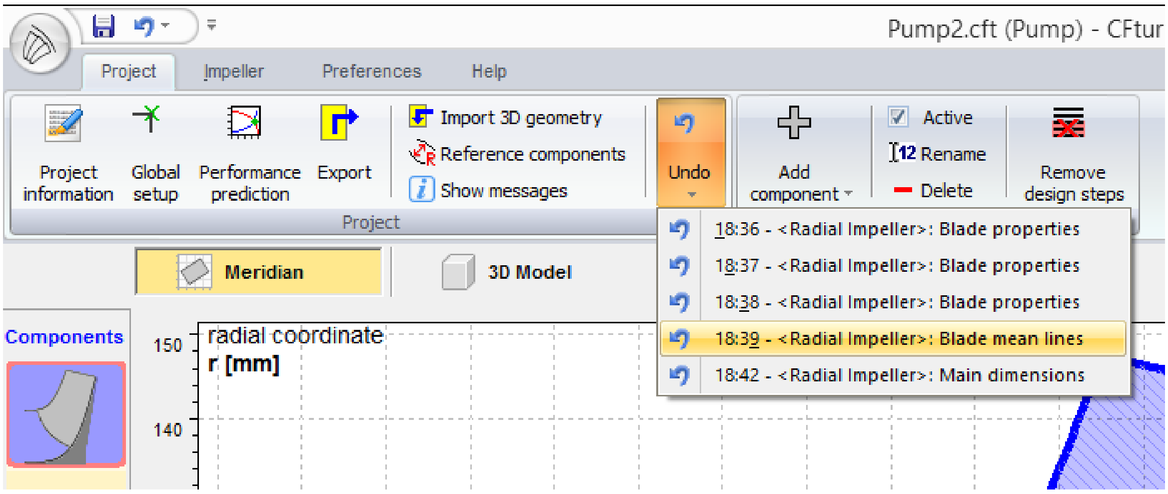
More information to the message panel are available in the Opened project 57 section.

6.2.1.8 Undo

? Project | Project | 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.

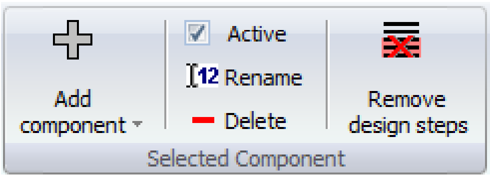


The undo-button is also placed in the quick access bar by default.

6.2.2 Selected component

? Project | Selected Component

All operations in this group refer to the currently selected component.



→ Add component 140

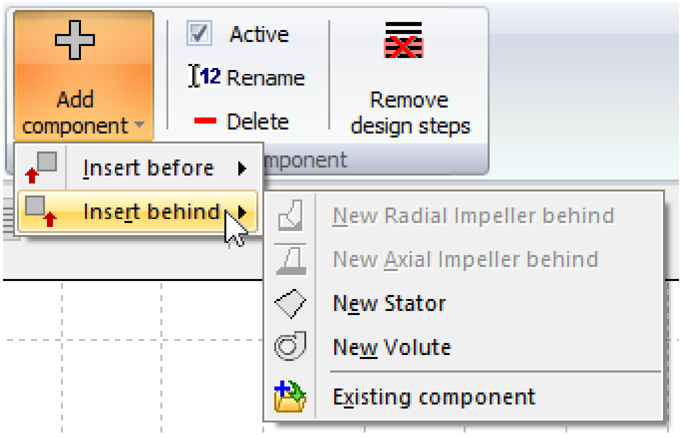
→ Active/ Rename/ Delete 141

→ Remove design steps 143

6.2.2.1 Add component

? Project | Selected Component | Add component

A new component can be inserted before or behind the currently selected one, followed by selecting the type or adding an existing one from another project.



There can be up to 2 impellers in a project and a single volute only.

An impeller can be added only if the flow direction on the selected position is suitable to the impeller geometry.

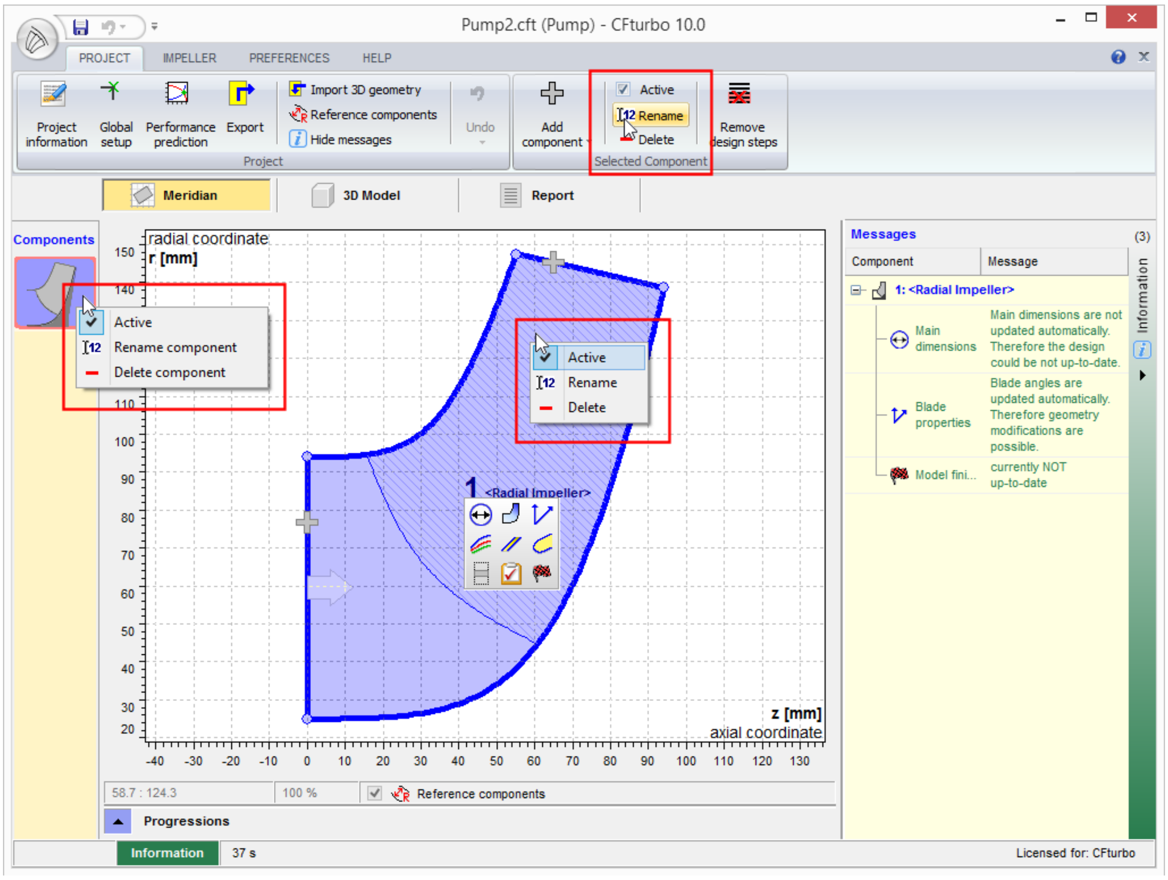
Alternatively you can add components in the Meridian view using the buttons between neighboring components (see Meridian 168).

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 71 are applied for this new component.

6.2.2.2 Active/ Rename/ Delete

The actions **Active**, **Rename** and **Delete** can be executed in the following manner alternatively:

- Menu **Project | Selected Component**
- Context menu of the corresponding component left in panel **Components**
- Context menu of the corresponding component right in the meridional preview



? Project | Selected Component | Active

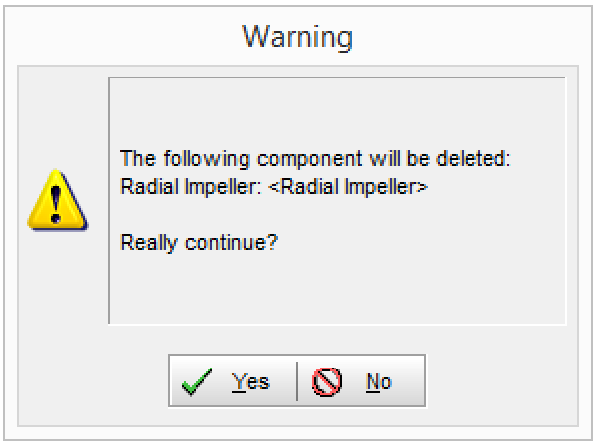
A inactive component is read only and also not going to be updated automatically. Inactive components are colored grey in all views.

? Project | Selected Component | 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.

? Project | Selected Component | Delete

The selected component is going to be deleted after confirming the warning.



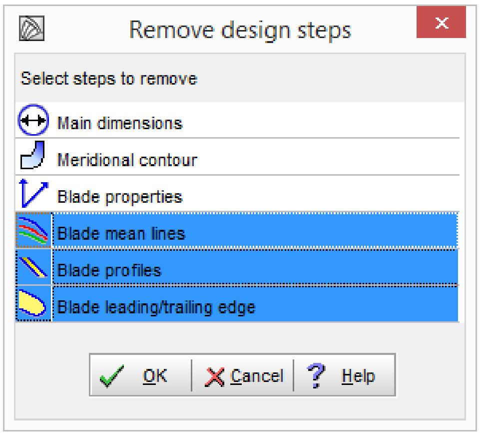
6.2.2.3 Remove design steps

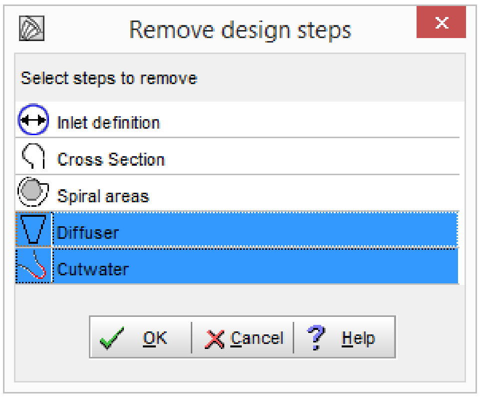
? Project | Selected Component | Remove design steps

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.



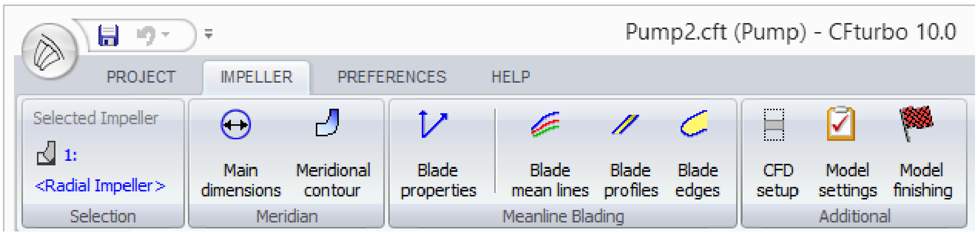


6.3 IMPELLER/ STATOR/ VOLUTE

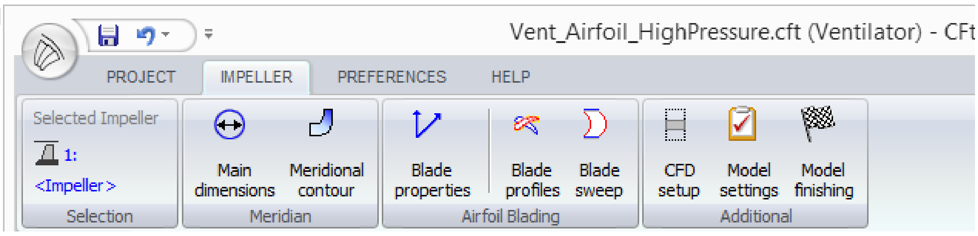
These menus are used for the actual component design.

A separate tab with the corresponding design steps is available for each component type:

IMPELLER 189

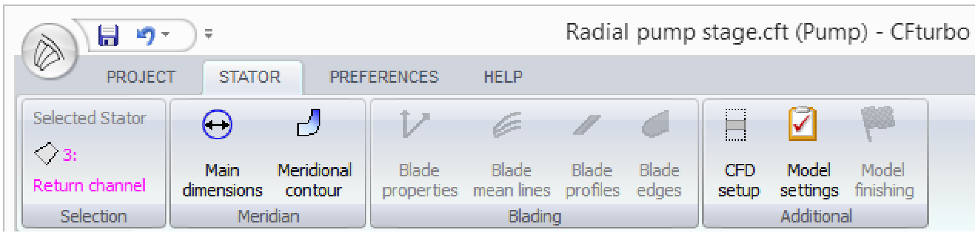


(Mean line 292 design mode 292)

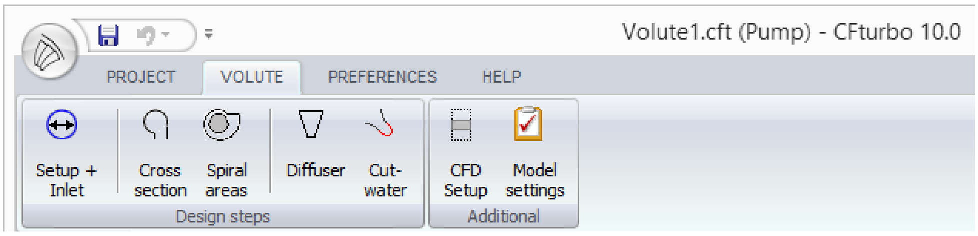


(Airfoil/ Hydrofoil design mode 351)

STATOR 384



VOLUTE 400



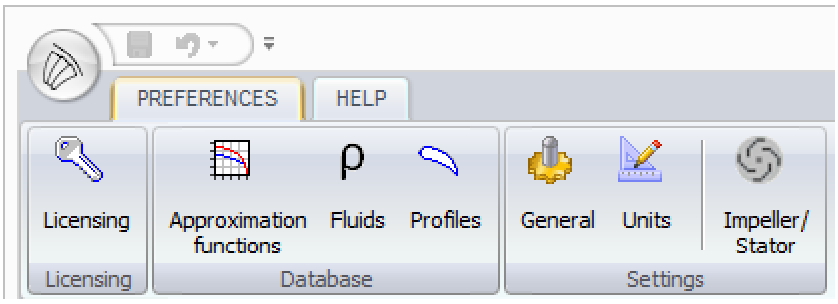
Menu items and buttons only become active in accordance to the current design state. Each finished design steps can be opened again whereas all depending design steps and components are updated automatically. Manual removing of complete component's design steps is possible in order to continue with CFturbo® initial design (see Remove design steps 143).

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 168).

6.4 PREFERENCES

This menu is used for specifying some general program settings:



- Licensing 145
- Approximation functions 145
- Fluids 148
- Profiles 152
- General 155
- Units 158
- Impeller/ Stator 161

6.4.1 Licensing

? Perferences | Licensing | Licensing

See General/ Licensing 12

6.4.2 Approximation functions

? Perferences | 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 116 functions are available for the following individual component types and sub-types:

- 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

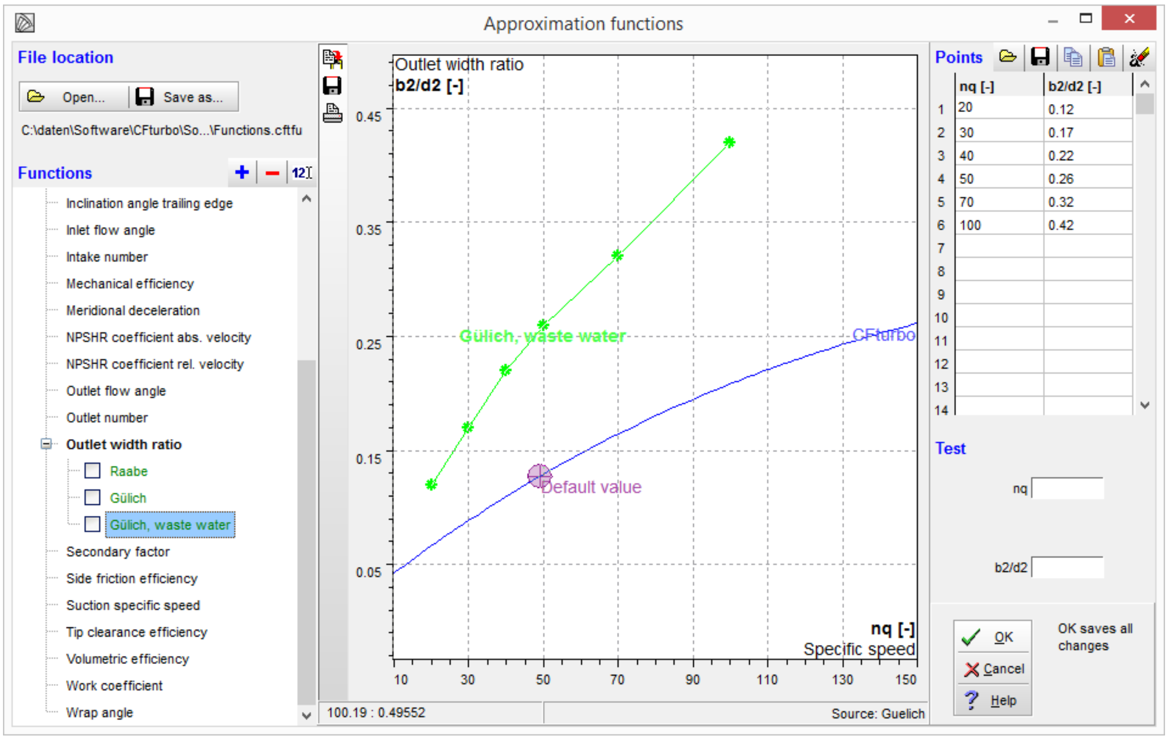
Each function has a hard coded default function. For each of these functions custom point wise defined curves can be added alternatively. These custom defined curves are saved in the file **Functions.cftfu** that contains the custom defined functions only. The default functions are not saved in any external file and cannot be deleted. The default functions can only be deactivated by defining any custom function that is saved in the Functions file.

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

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

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

- all users can use the same database for their design
- there is no risk of losing data by uninstall older versions of CFturbo

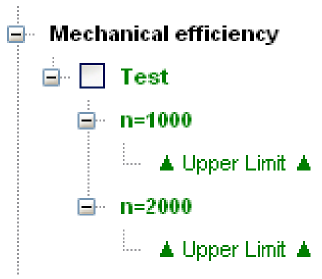


All available functions are listed in a tree structure in the panel **Functions** left from diagram, sorted by machine type.

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

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

The following hierarchy exist in the tree:



physical variable

user defined function

parameter curve (available for some functions only)

upper limit curve (optional)

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:

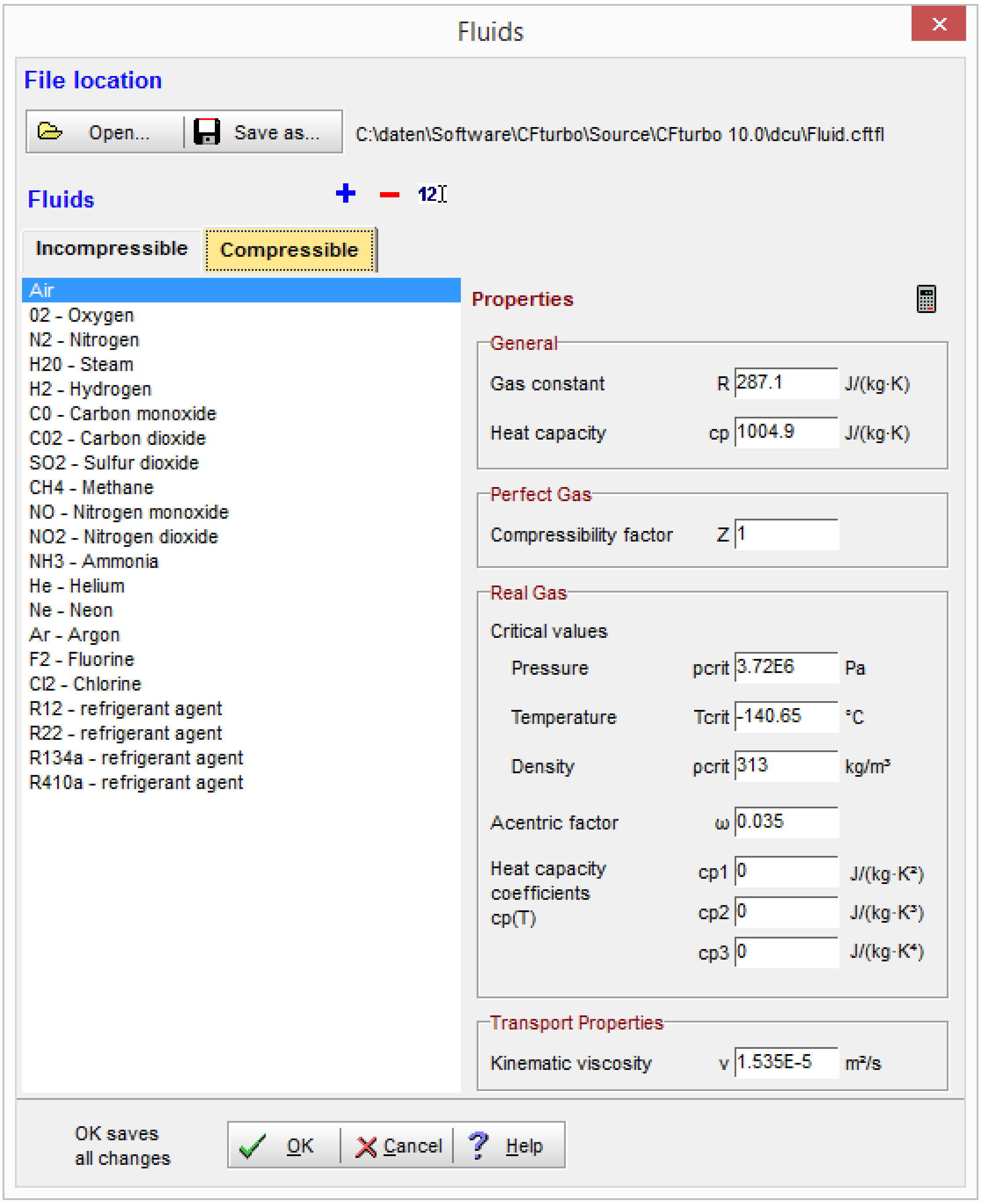
- import points from file (one point per line)
- export points to file
- copy all points to clipboard
- paste points from clipboard (e.g from Excel)
- clear the table

On panel **Test** you can test the active function. Saving of values is possible by clicking **OK**-button.

6.4.3 Fluids

? Preferences | Database | Fluids ρ

The dialog lists all defined fluids. New ones can be added, present fluids can be 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.

Incompressible fluid [for pumps, ventilators only]

A constant density ρ is the only parameter.

Compressible fluid [for compressors, turbines only]

In this case 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
- critical pressure p_{crit}, temperature T_{crit} and density ρ_{crit}
- acentric factor ω
- heat capacity c_p + heat capacity coefficients c_{pi} (both at zero pressure)
- compressibility factor Z

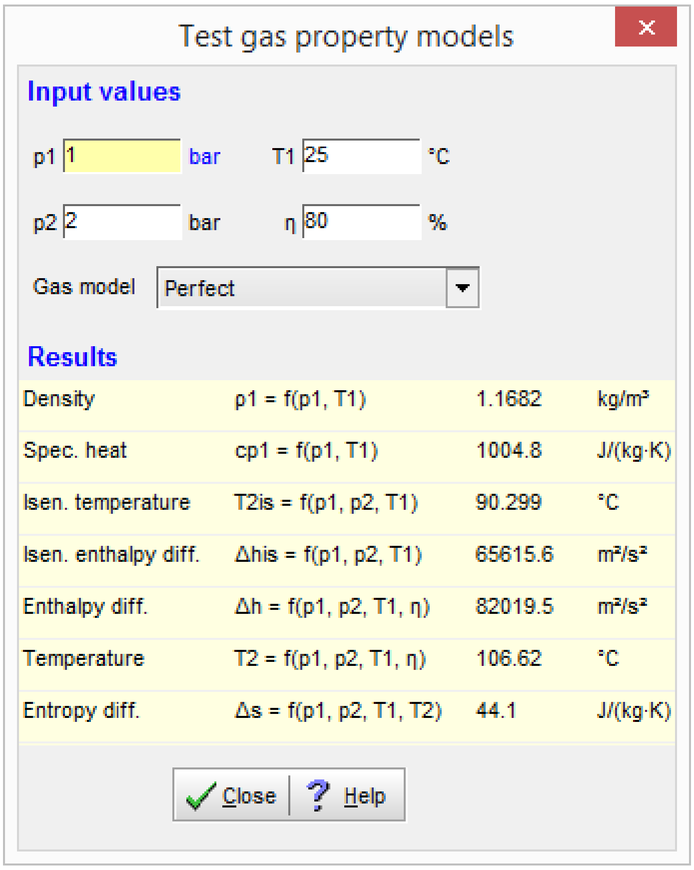
Currently the following gas models are implemented. They represent a relation between pressure, temperature and density (here given with its reciprocal the spec. volume v):

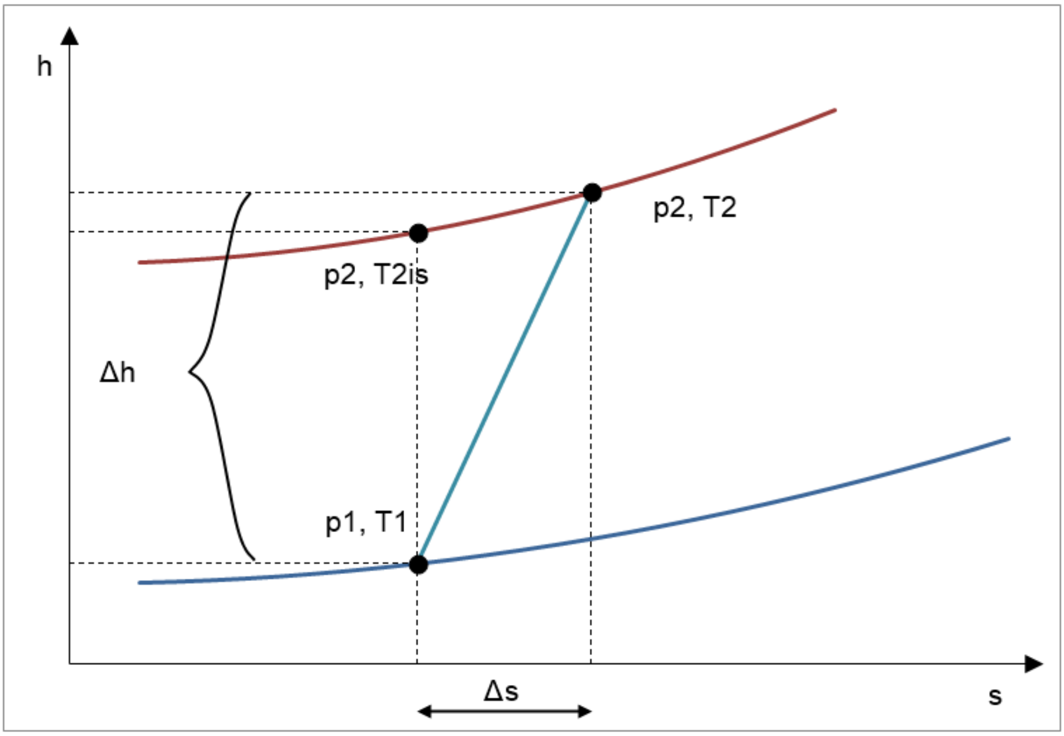
Gas model	Approach	Annotation	Reference (first published)
Perfect Gas	$p = \frac{R \cdot T \cdot Z}{v}$		
Redlich-Kwong	$p = \frac{R \cdot T}{v - b + c} - \frac{a(T)}{v(v+b)}$	Each approach has its own set of coefficients a, b and c.	Redlich, O., Kwong, J.N.S. 451
Aungier/Redlich-Kwong			Aungier, R.H. 451
Soave/Redlich-Kwong			Soave, G. 451
Peng-Robinson	$p = \frac{R \cdot T}{v - b + c} - \frac{a(T)}{v^2 - 2vb + b}$		Peng, D.Y., Robinson, D.B. 452

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 Δh and Δs resp. is given too, see h-s-diagram.



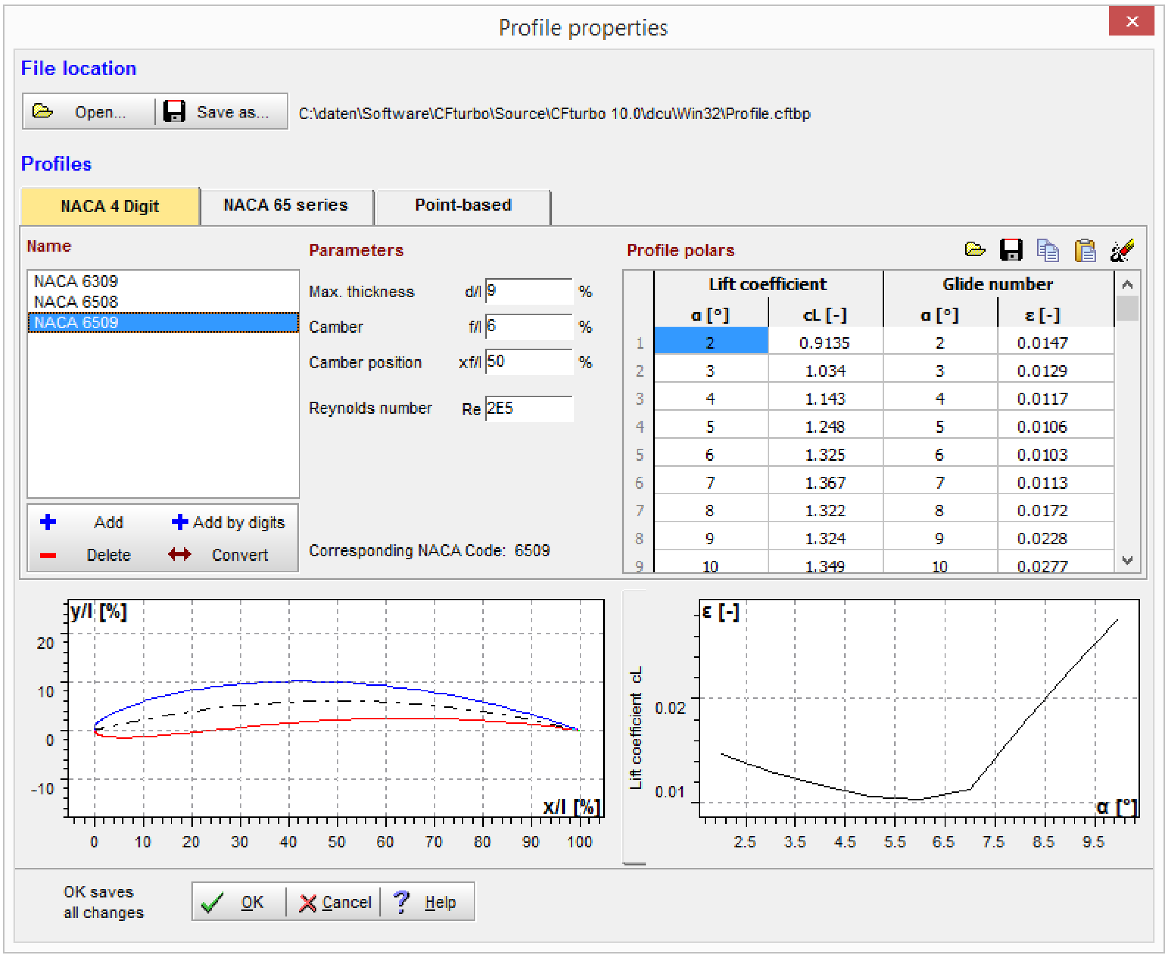


6.4.4 Profiles

? Preferences | Database | Profiles

[Axial machines only]

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 four-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:

$$\frac{y_d}{d}=\frac{1}{0.2}\left[0.2969\left(\frac{x}{l}\right)^{0.5}-0.126\left(\frac{x}{l}\right)^{1}-0.3516\left(\frac{x}{l}\right)^{2}+0.2834\left(\frac{x}{l}\right)^{3}-0.1015\left(\frac{x}{l}\right)^{4}\right].$$

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

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

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

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 449 p. 54, (Eq. 3.11, 3.12):

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

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

Nose radius and thickness can be modified.

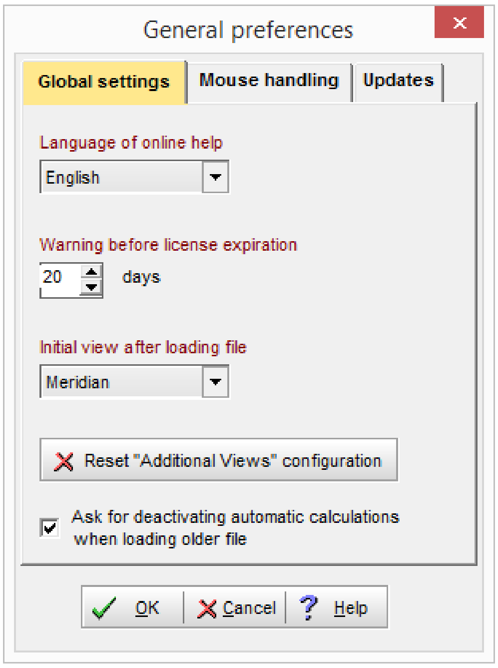
Point-based

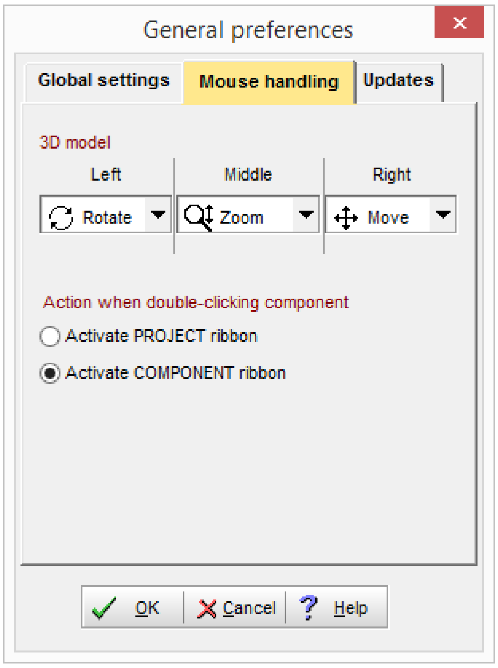
Besides NACA profiles also user-defined profiles are provided. Therefore the lower and upper side of the profile has to be known. Moreover lift coefficients and glide numbers need to be set with respect to the angle of attack.

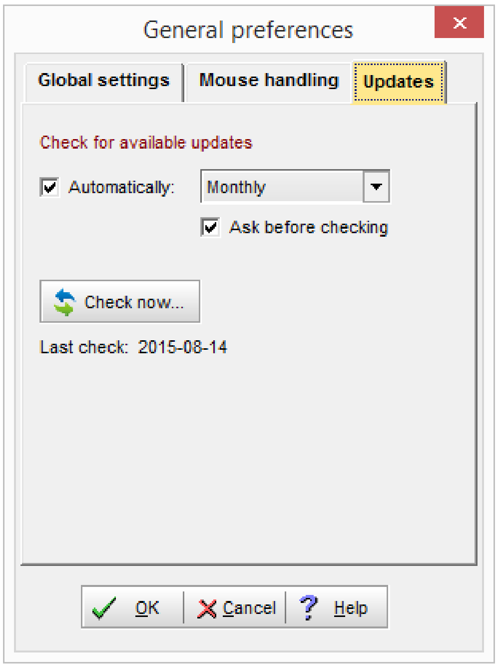
6.4.5 General

? Preferences | Settings | General

Menu item **General preferences** is used for global program options.







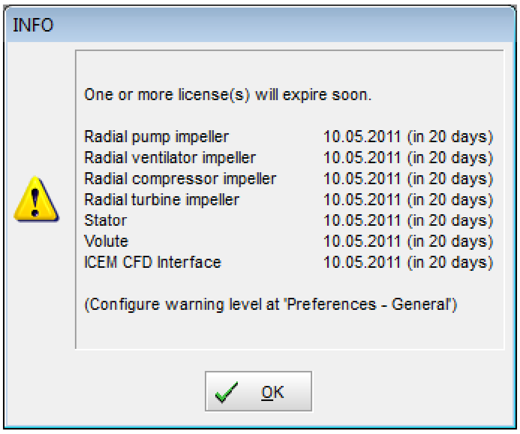
Language of online help

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 42.

3D model mouse handling

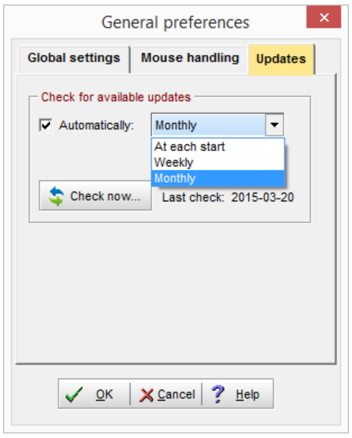
Here you can assign functions (Rotate, Zoom, Move) to the mouse buttons (Left, Middle, Right) for handling the 3D model 172.

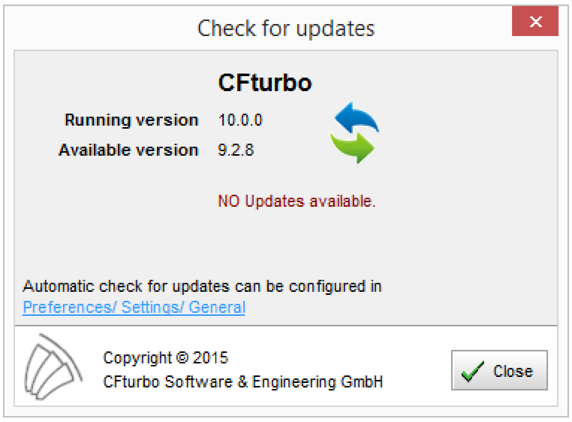
Action when double-clicking component

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 164). The date of last update check is displayed for information.

6.4.6 Units

? Preferences | Settings | Units

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

It's divided in 3 parts:

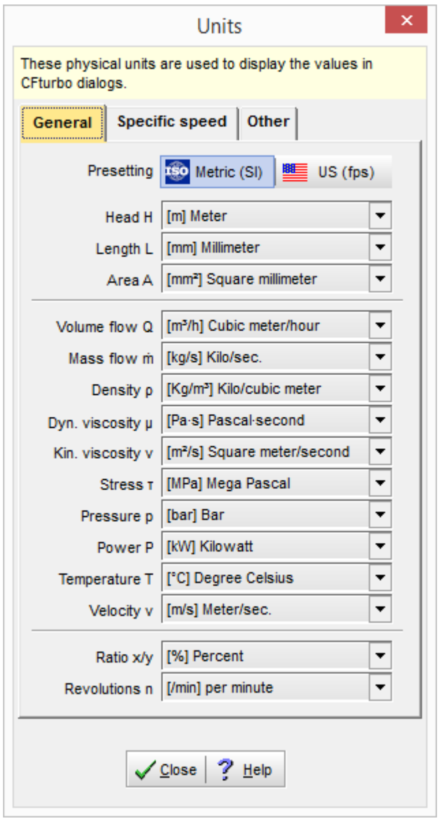
➜ General158: general unit selection

➜ Specific speed159: selecting a suitable specific speed definition

➜ Other160: some additional unit settings, like flow/blade angle and n_{ss} definition

6.4.6.1 General

Here the physical units used in the dialogs can be set. Following units are available:



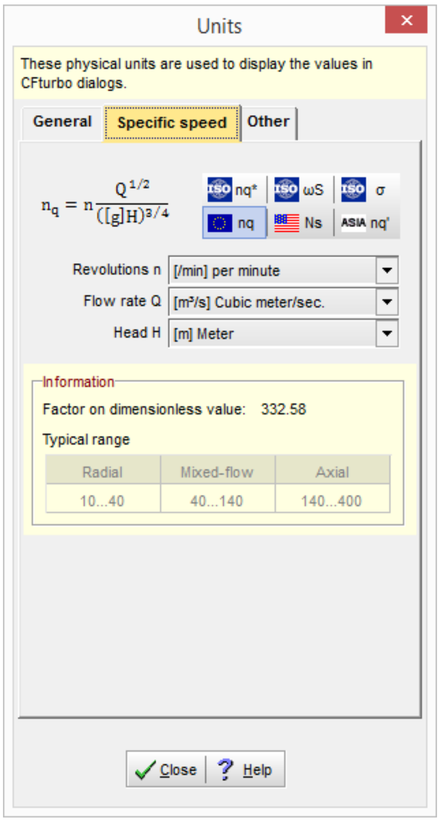
- Head: m, ft
- Length: mm, in, m
- Volume flow: m^3/h, m^3/min, m^3/s, ft^3/h, ft^3/s, gpm, gps
- Density: kg/m^3, lb/ft^3
- Stress: MPa, PSI
- Pressure: MPa, PSI, bar, Pa, mm H_20, in H_20, ft H_20
- Power: kW, hp
- Mass flow: kg/s, lb/s
- Temperature: °C, K, °F
- Area: mm², m², in²
- Velocity: m/s, ft/s
- Dynamic viscosity: Pa·s, cP
- Kinematic viscosity: m²/s, ft²/s
- Ratio: %, -
- Revolutions: /min, /s

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

6.4.6.2 Specific speed

Here the specific speed definition can be selected. This definition is mainly used for the Approximation functions 145.

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



Following definitions are available:

- General specific speed n_q^* (dimensionless)

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

- Type number ω_s (dimensionless)

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

- Speed coefficient σ

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

- European definition n_q

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

- US definition N_s

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

- Asian definition n_q'

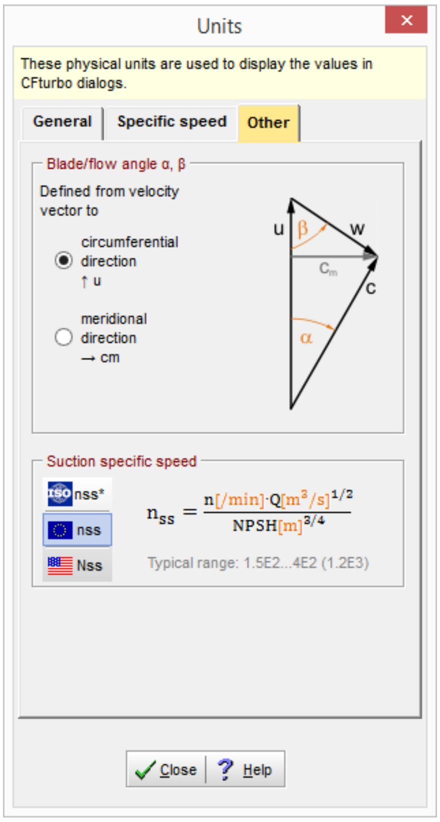
$$n_q = n[\mathrm{min}^{-1}]\frac{Q[m^3/\mathrm{min}]^{1/2}}{H[m]^{3/4}}$$

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

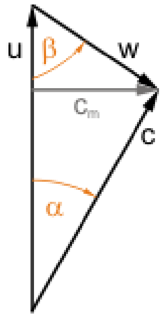
On the bottom side some information for the currently selected specific speed definition is displayed. The **Factor on dimensionless value** is the factor used to convert the General specific speed n_q^* to the currently selected definition. Furthermore the **Typical range** of the specific speed definition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

6.4.6.3 Other

Here some additional unit settings can be selected.



Blade/flow angle α, β

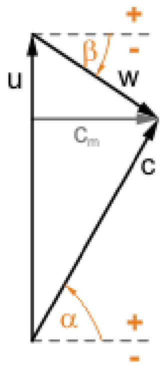


↑ u

angles measured against circumferential direction

(internal angles of the velocity triangle)

allowed range: 0° … 180°



→ cm

angles measured against meridional directi

allowed range: -90° … +90°

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}

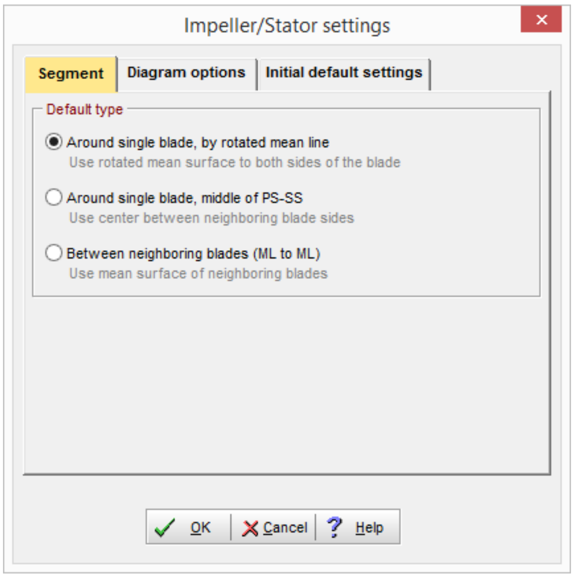
- US definition N_{ss}

$$N_{ss} = n[rpm]\frac{Q[gpm]^{1/2}}{NPSH[ft]^{3/4}}$$

6.4.7 Impeller/ Stator

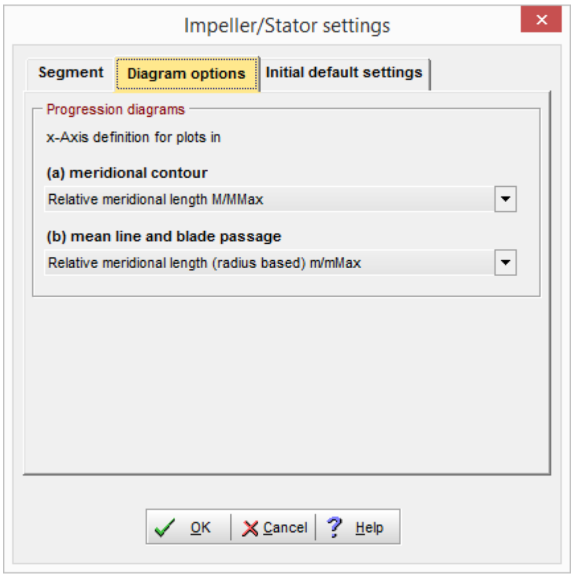
? Preferences | Settings | Impeller/ Stator

Menu item **Preferences - Impeller Options** is used for global default definition. These settings are set at the initial opening of each dialog.



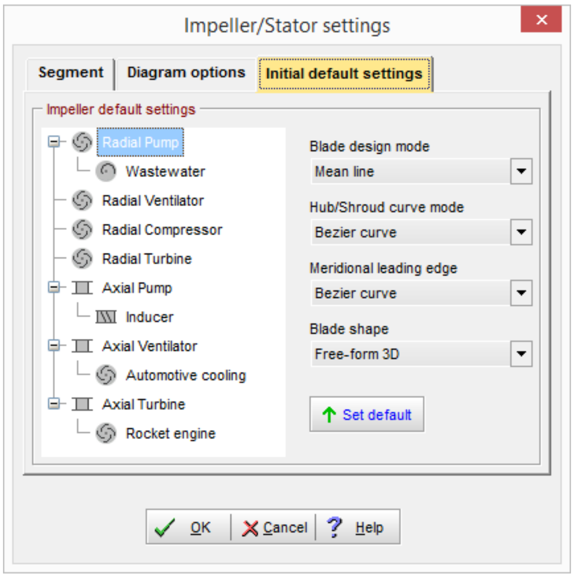
On tab sheet **Segment** the default position of the rotationally symmetric blade segment can be selected.

Detailed information is available at the CFD setup 371.



On tab sheet **Diagram options** one can specify, which parameter should be used for the x-axis of the progression diagrams in the Meridional contour 268 and Mean line 319 dialog as well as for the cross section 178. Some constellations may yield undefined x-values due to reference (e.g. r_{Max}, Δz_{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 $\Delta z/\Delta z_{Max}$

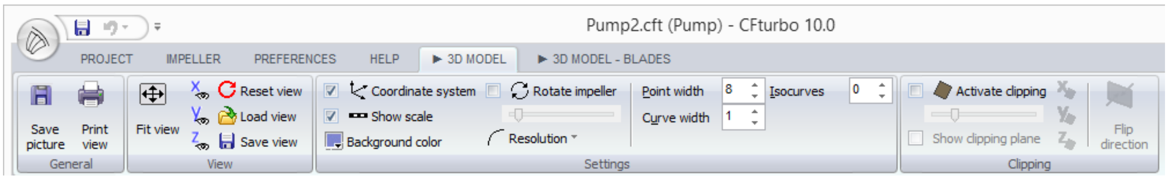


On tab sheet **Initial default settings** one can select which settings should be used by default when creating a new design. Individual settings can be specified for each machine type (Pump, Ventilator, Compressor, Turbine).

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

6.5 3D MODEL

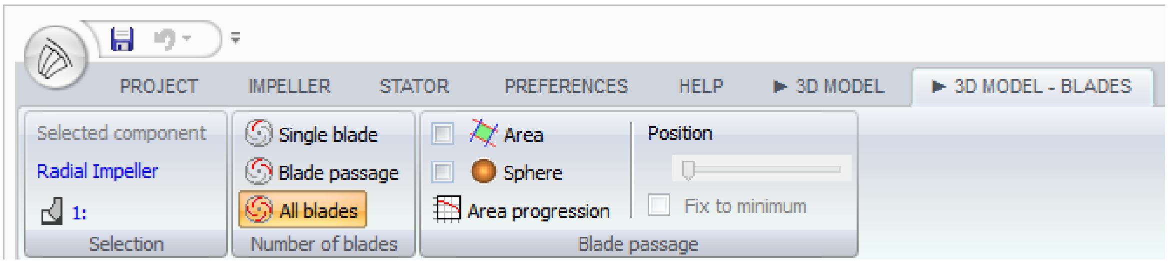
This Menu is used for general handling of the 3D model.



Detailed description can be found in Views/ 3D Model 172.

6.6 3D MODEL - BLADES

This Menu is used for handling geometries with blades (impeller, vaned stator) in the the 3D model.

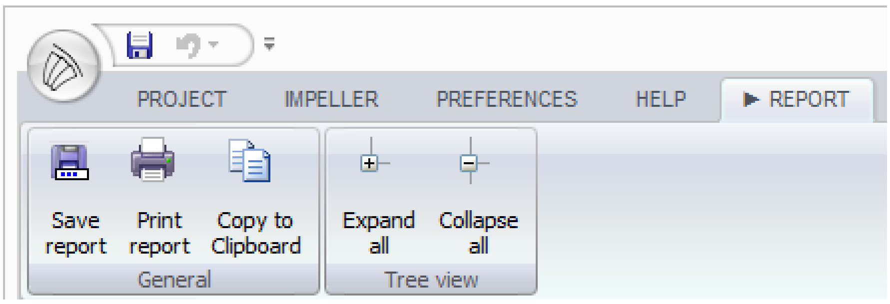


Because a project can contain multiple geometries with blades, these settings refer to the currently selected component in the model tree 179 of the "3D Model" view. The name of the selected 3D component is displayed for information leftmost in the menu.

Detailed description can be found in Views/ 3D Model 172.

6.7 REPORT

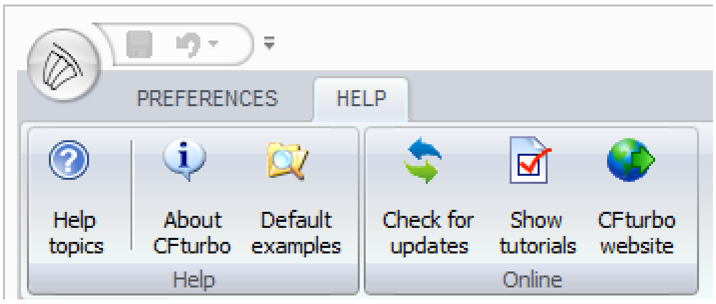
This Menu is used for handling the project report.



Detailed description can be found in Views/ Report 186.

6.8 HELP

This menu supports the user on how to use CFturbo.



The following features are available:

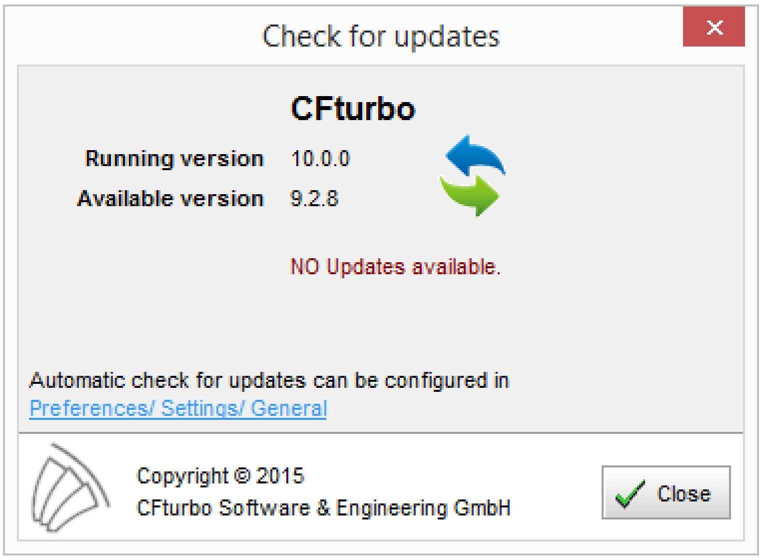
	Help topics	General CFturbo online help, including help index
	About CFturbo	Information about CFturbo (e.g. version information)
	Default Examples	Open default examples folder of CFturbo installation
	Check for updates 164	Check for updates online
	Show tutorials	Show online tutorials for CFturbo
	CFturbo website	Open CFturbo website in browser

6.8.1 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 10.0.x 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 Preferences/ General 155.

Part VII

7 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 65.

- Meridian 168

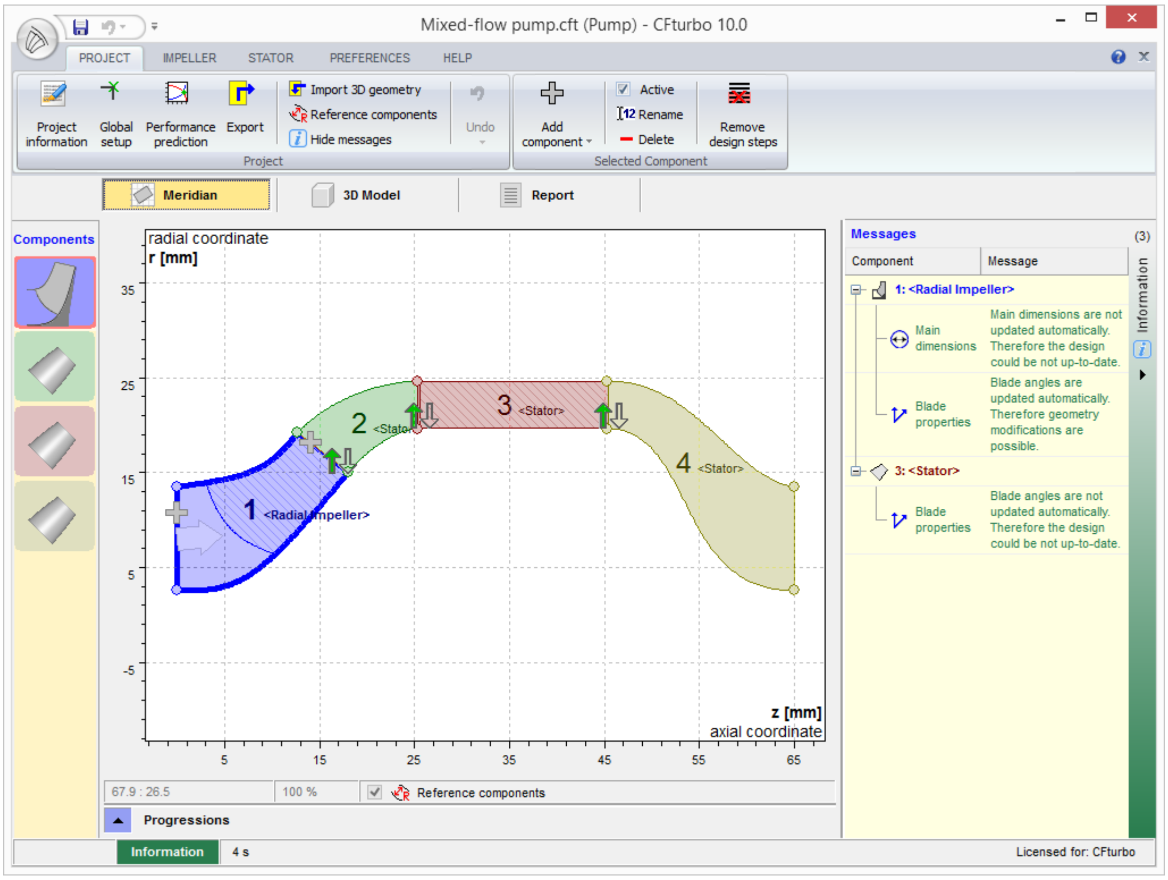
  The diagram with the meridional view of the components gives an overview of the project and enables quick access to the components and the Interfaces 38 in between.

- 3D Model 172

  Shows the whole project as a 3D model.

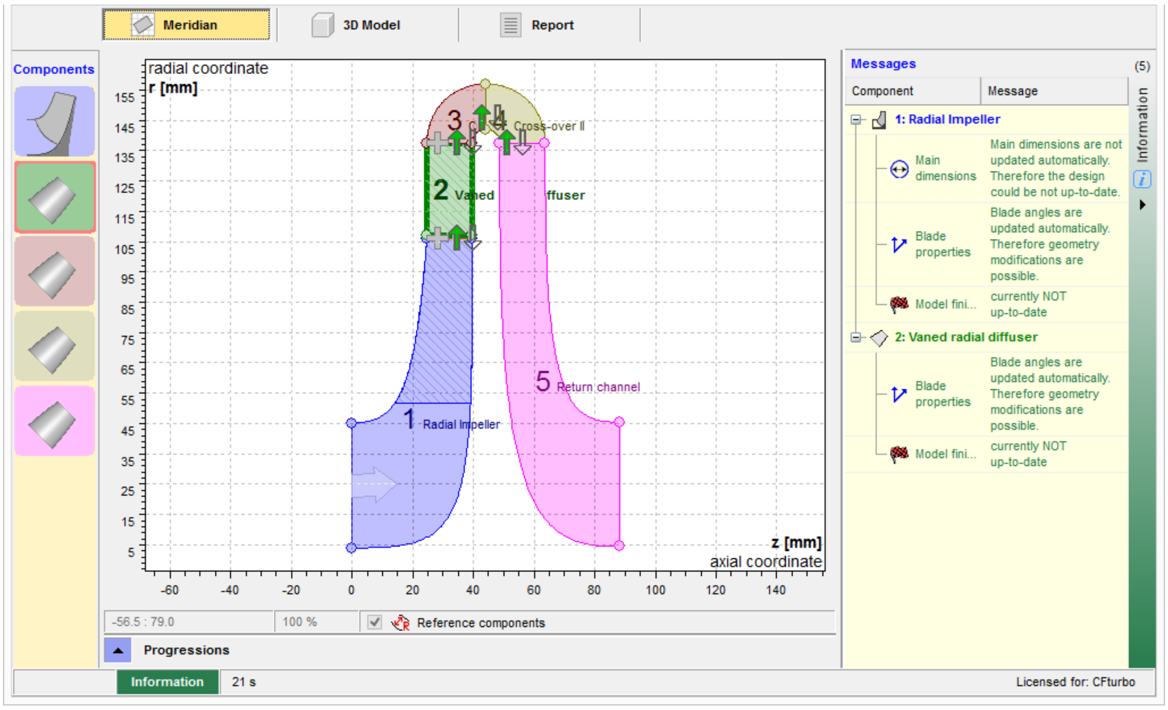
- Report 186

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



7.1 Meridian

This view consists mainly of a diagram containing the meridional shape of all components.

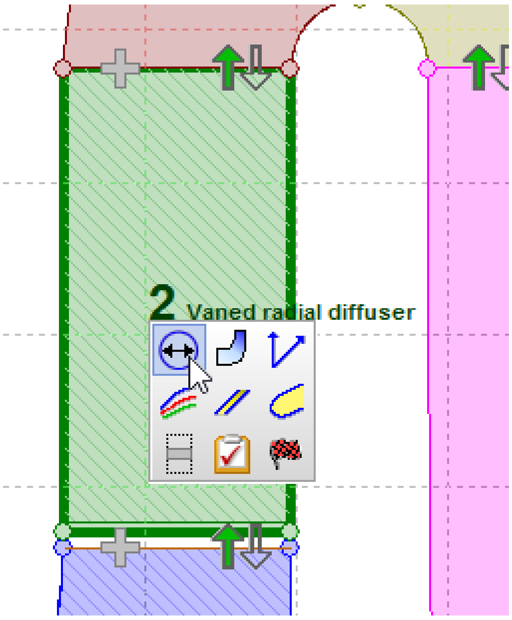


Active components are displayed with their respective color, inactive components are displayed grey.

Meridional diagram

The diagram depicts the assembled meridional shapes of the project components and their connecting interfaces
A large arrow on the inlet of the first component illustrate the flow direction.
Captions showing component name and a consecutive number are displayed as well.

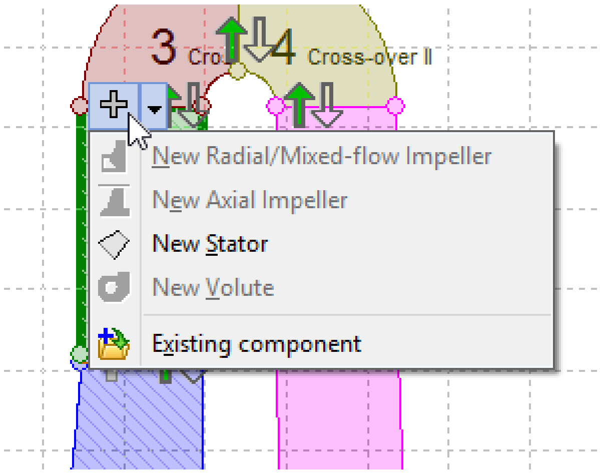
The currently selected component is displayed with thick border and can be changed by mouse click on a component.



Component context menu

If the mouse moves over the selected component the components menu is shown in compact style.

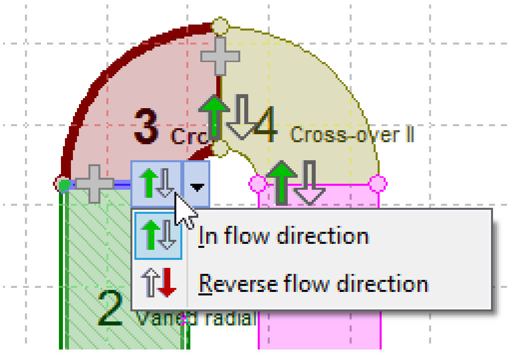
Alternatively you can use the corresponding ribbon menu (see IMPELLER/ STATOR/ VOLUTE 144).



Adding Components

Via the ✚ symbol an additional component can be added to the project at the symbols position.
A menu shows the available component types and the option to import an existing one.

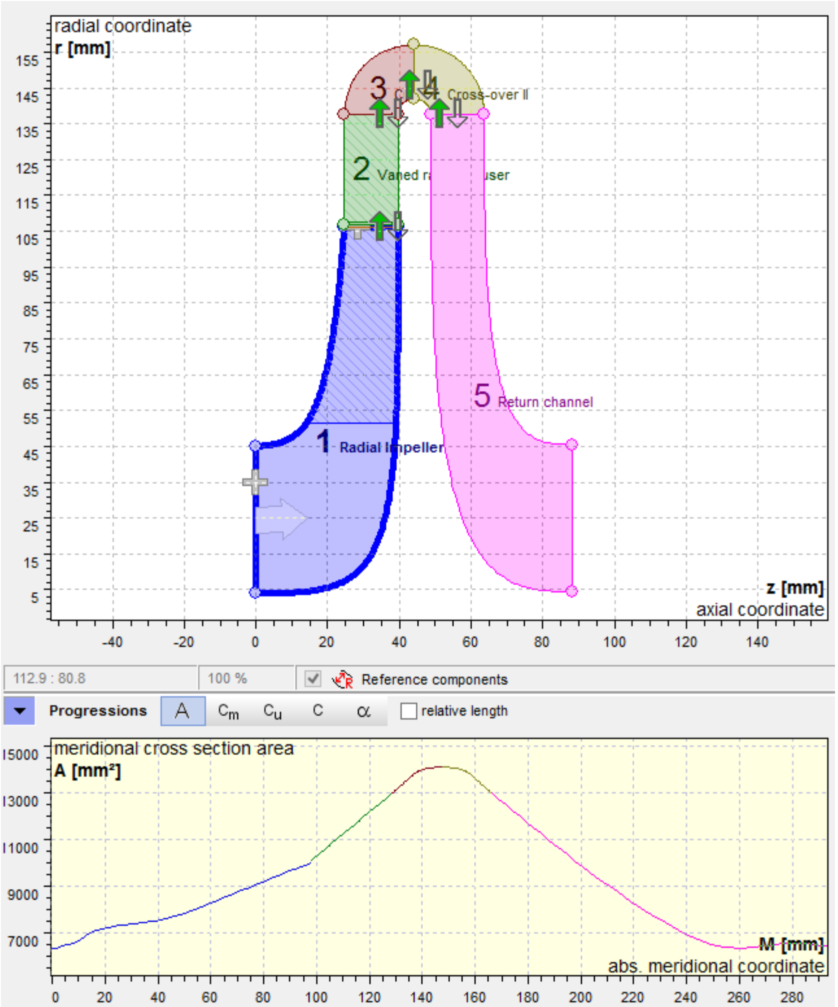
Alternatively you can use the corresponding ribbon menu (see Add component 140).



Interface coupling

Interfaces 38 are located between components. The direction of the interface coupling 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



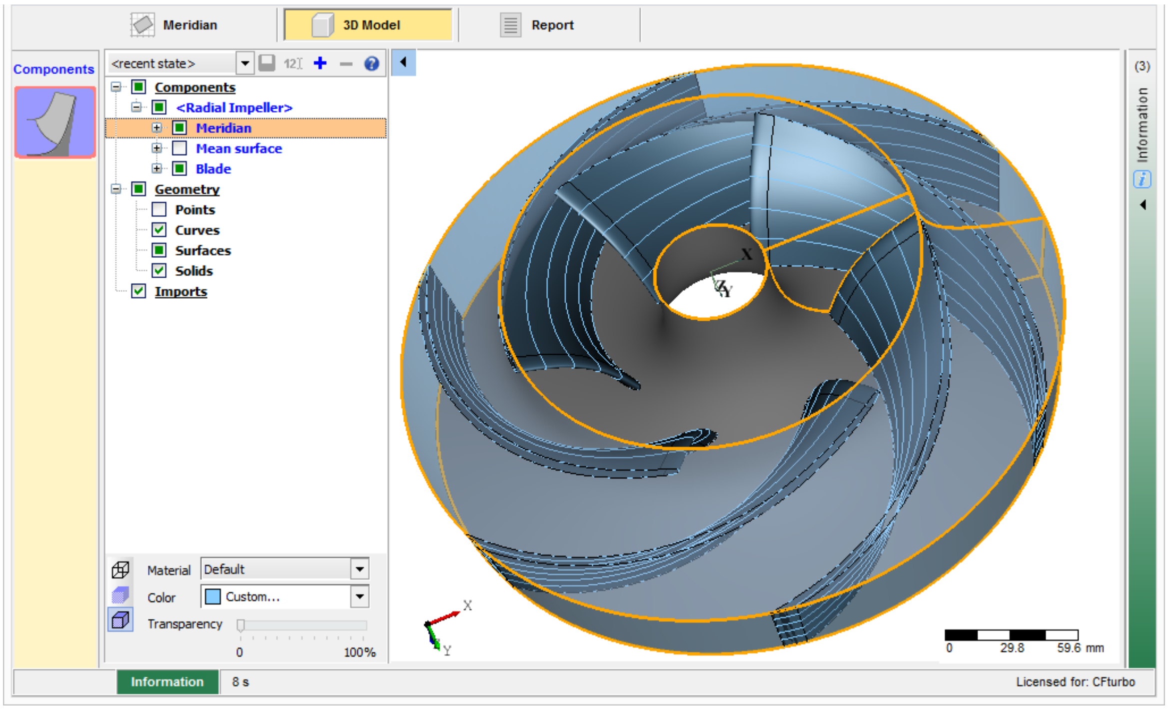
Below the meridional view, **progressions** of several physical quantities along the flow direction of all components can be displayed:

A	Cross section area
c_m	Meridional velocity
c_u	Circumferential velocity
c	Absolute velocity
α	Flow angle

7.2 3D Model

Tab sheet **3D Model** contains the three dimensional representation of the project design state.

The CAD model can be exported as IGES, STEP or STL - see Export 85. For export, only the currently visible geometrical elements are considered.



Navigation

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

Rotate	Rotation around point of origin
Zoom	↕ Zoom (also mouse wheel) ↔ Rotation around z-axis
Move	Move

The functions can be assigned to mouse buttons via Preferences/ General 155.

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) 173

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) 179

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 coarse resolution only.

See also:

➜ Problems when generating surfaces/solids 183

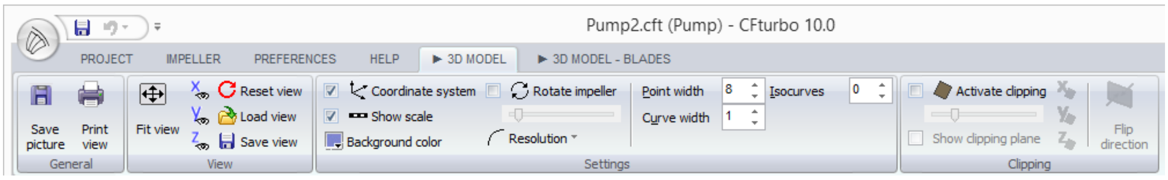
➜ Open/ Save design 69

➜ Data export 85

7.2.1 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

	Save representation as PNG, JPG, GIF or BMP
	Print representation

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

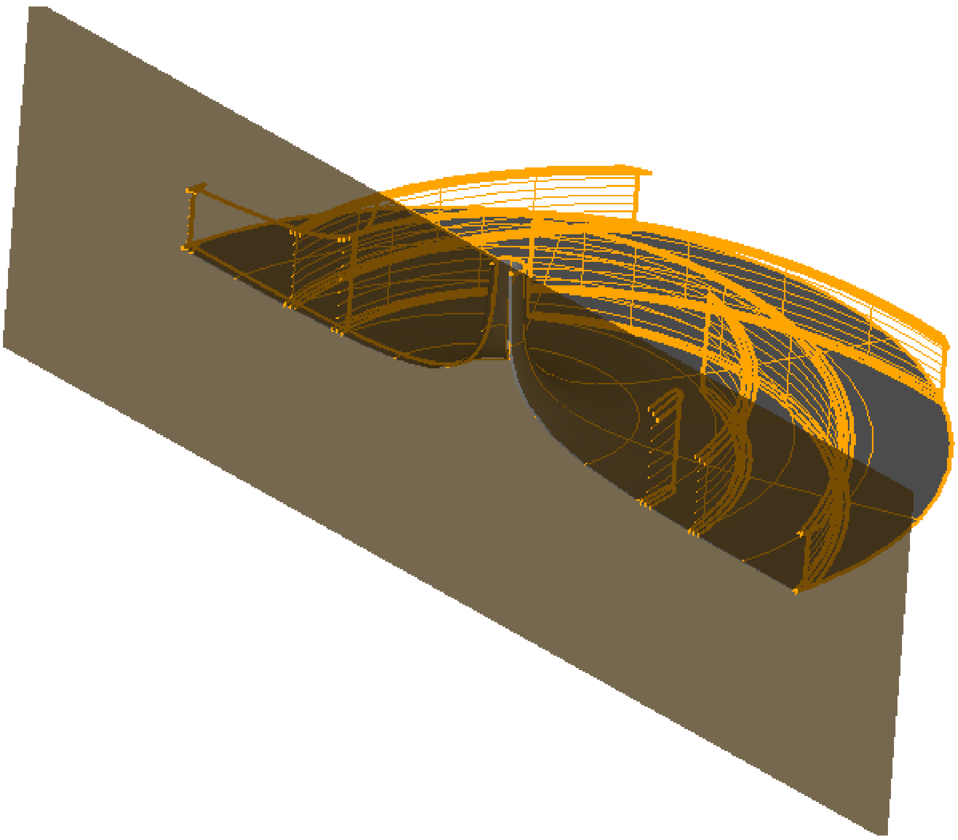
	Switch coordinate system on/off
	Switch scale system on/off
	Set background color
	A uniform rotation of the impeller around the z axis can be generated, whereby the velocity can be influenced by the track bar.

	Select resolution of curves and surfaces (affects display)
	Coarse
	Middle
	Fine
	Define line width for points Define line width for curves Set number of surface isocurves

Clipping

A clipping plane for x=const., y=const. or z=const. can be defined and optionally displayed. The position of the clipping plane can be adjusted by the track bar.

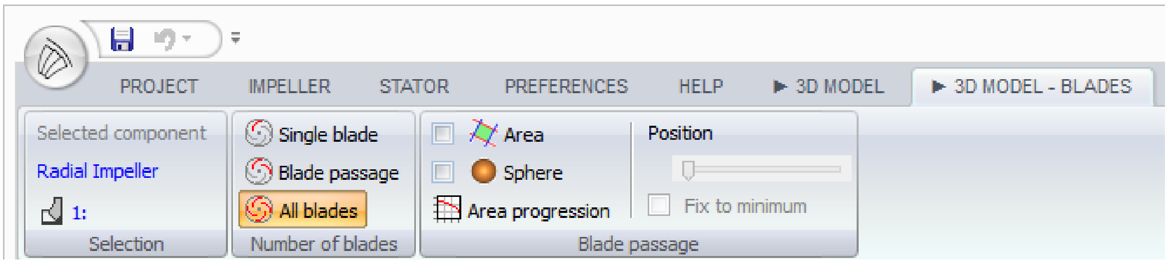
The direction of clipping (visible clipping side) can be switched.

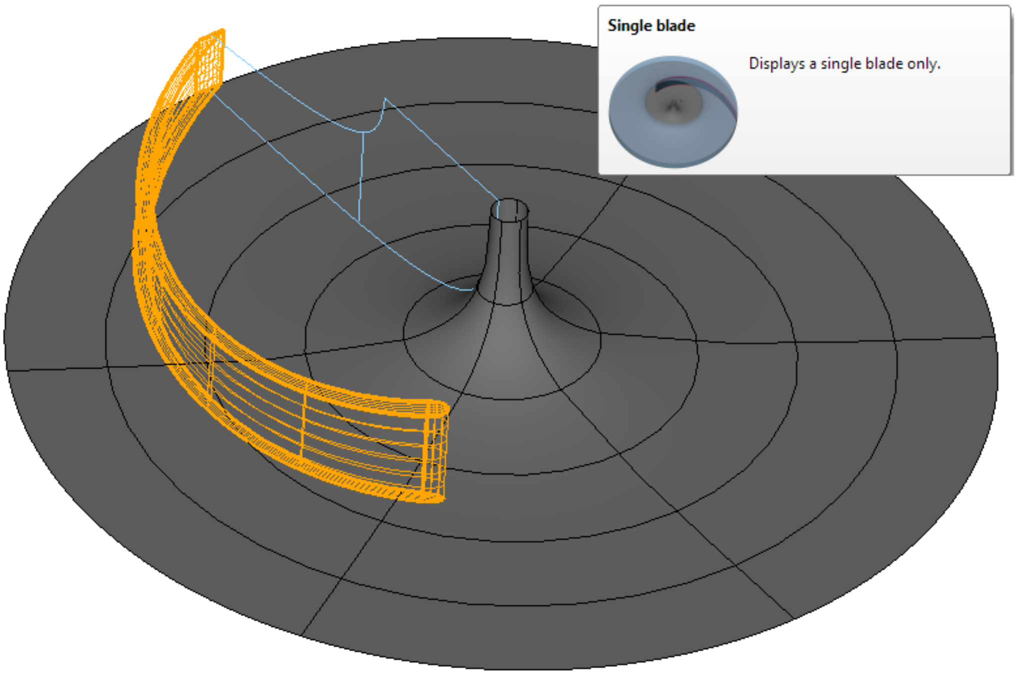


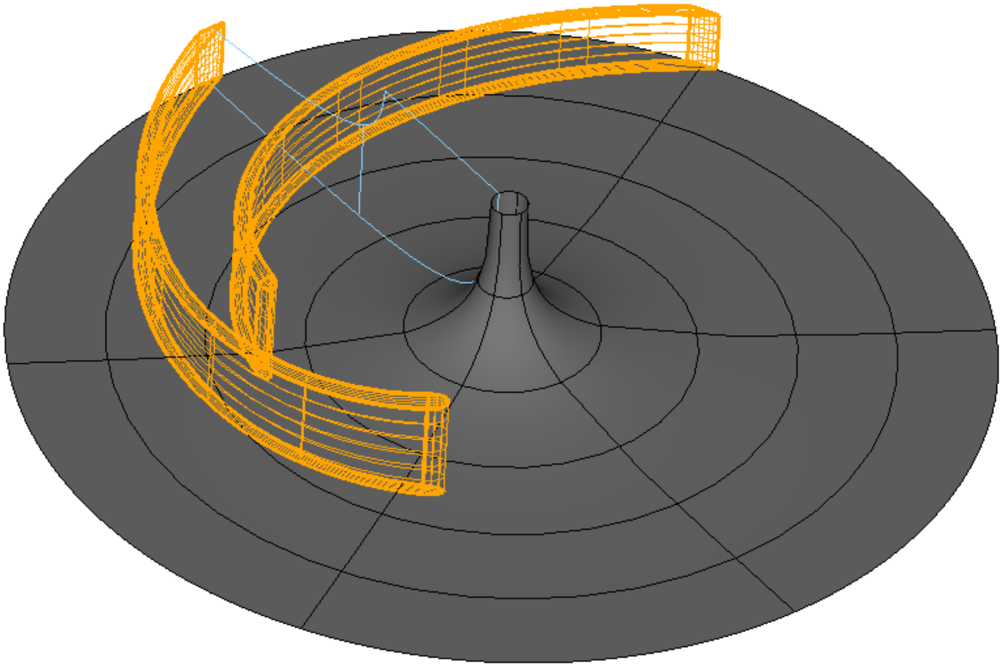
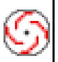
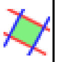
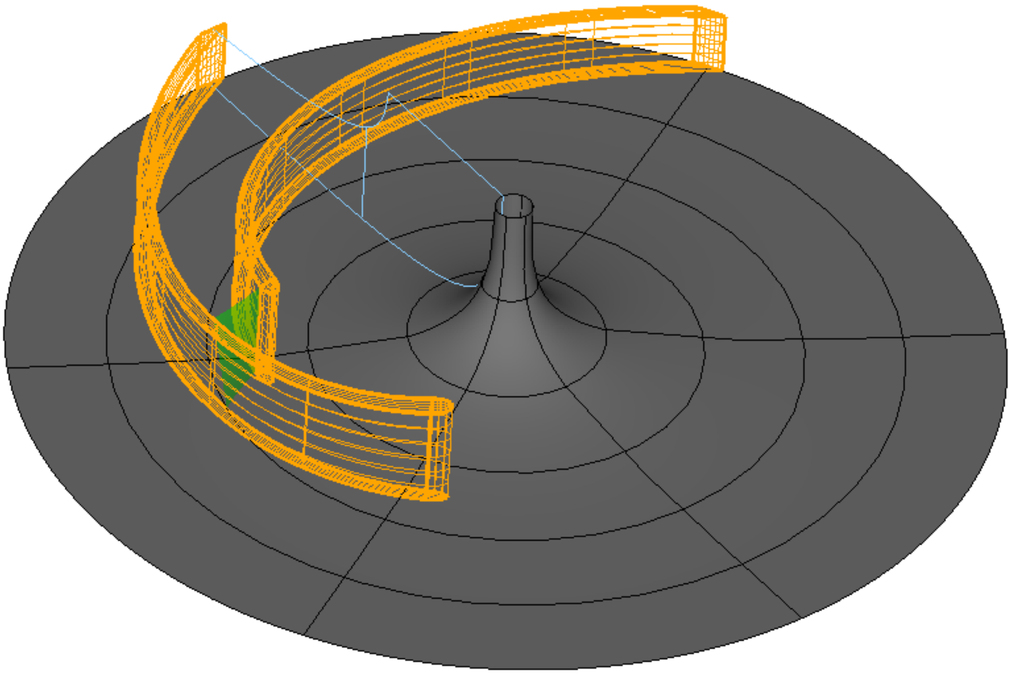
? 3D Model - Blades

The following actions are available through buttons of the **3D Model - Blades** tab. They 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.

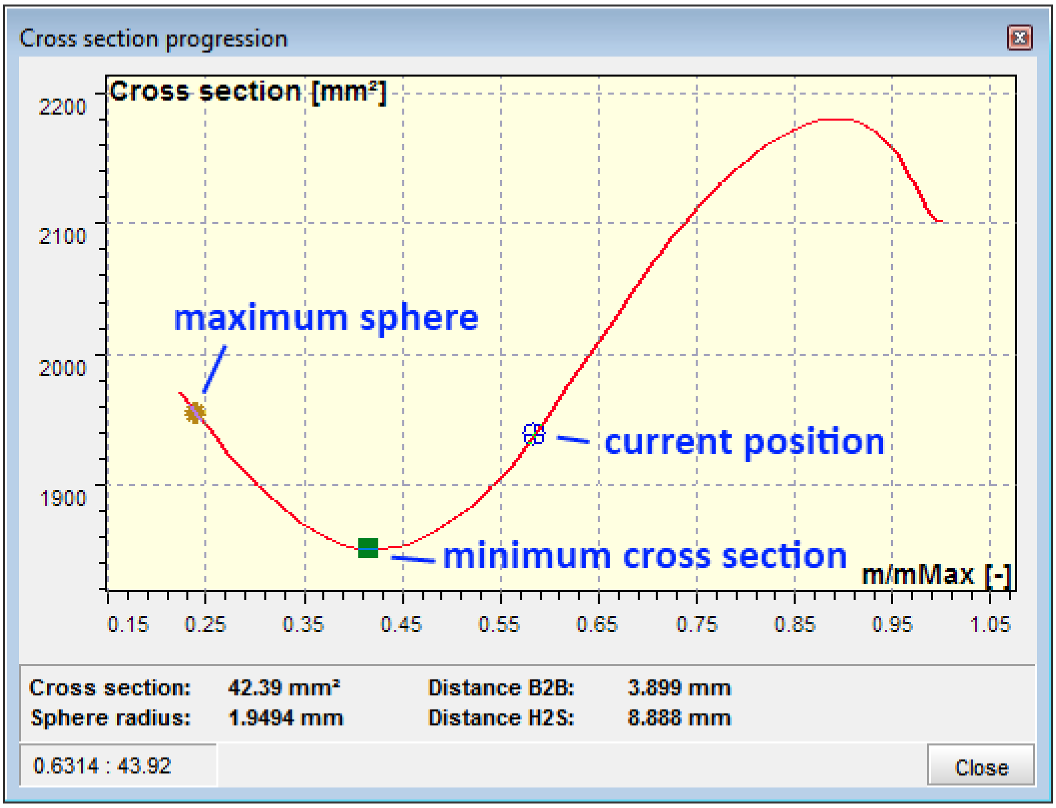


	Single blade 
	Blade passage Display a single blade passage bordered by 2 neighboring blades.

	
	All blades Display all blades of the selected impeller or vaned stator.
	Area 

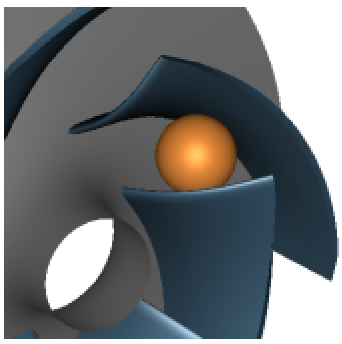
Display an approximately perpendicularly flown through area between hub, shroud and two neighboring blades for the currently selected component. The position of this area can optionally be fixed to the location of the throat area (**Fix to minimum**). Otherwise, it can be slided to any reasonable position within the blade to blade channel with the help of the track bar **Section Position**.

By pressing the button **Show progression** a window is opened, in which the value of the cross section is displayed in dependence on the position (see here 161 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.

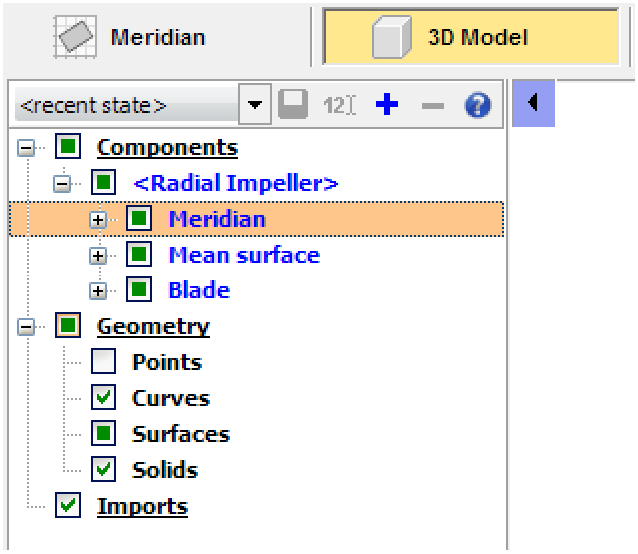


Sphere

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



7.2.2 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 or BREP - see Export 85.

Tooltips: If the mouse is paused over an item of the model tree its geometric parameters are displayed: volume (for solids), area (for surfaces) and length (for edges).

Model tree structure

The model tree has 3 main sections:

1) Section **Components**

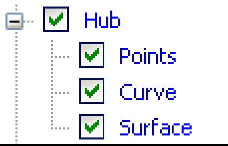
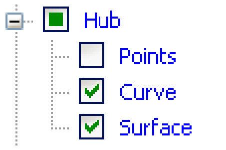
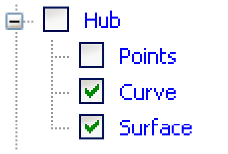
contains all components of the project with the following sub elements:

Impeller/Stator	Volute
Meridian	Spiral
Mean surface	Diffuser
Blade	Cut-water
CFD Setup 368	CFD Setup 444

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:

	The element and all child elements are selected.
	The element and not all child elements are selected.
	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.

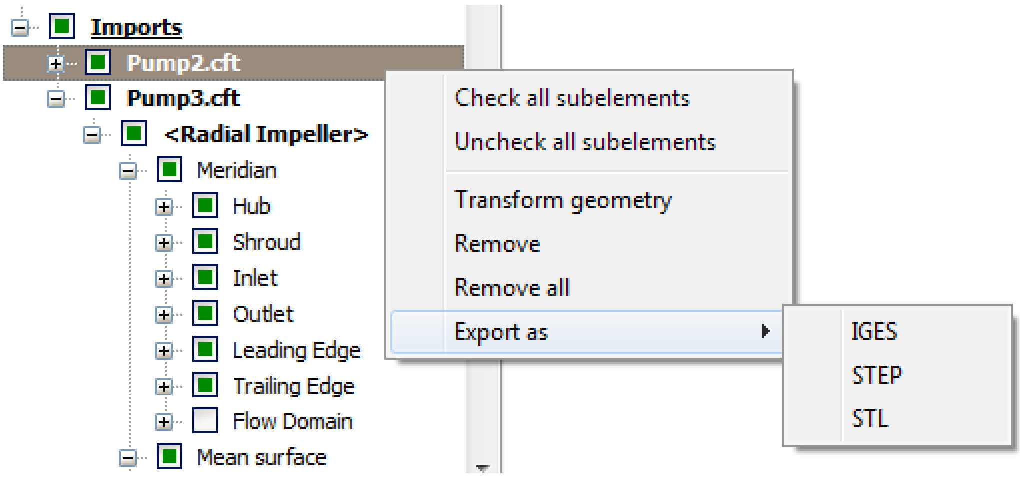
3) Section **Imports**

This section contains all imported geometric models including CFturbo components of reference projects 135 or simply imported 3D models 135.

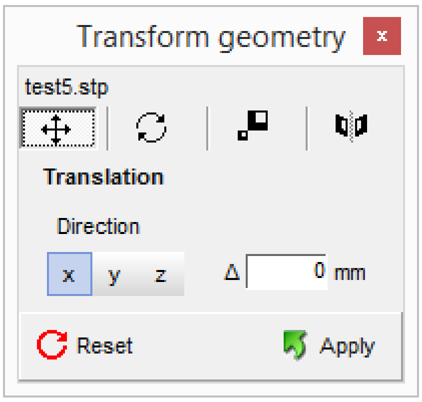
Visibility and render properties for imported models can be modified in the same way as for components of Section *Components*.

Right clicking on items in the *Imports* section provides a context menu with additional import related options:

- Transform geometry - applies user defined geometric transformations to currently selected import.
- Remove - removes selected import from model tree and 3D view.
- Remove all - removes all imported models from model tree and 3D view.
- Export as - exports selected import in its transformed state. (*This option is not available for STL imports*.)



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.

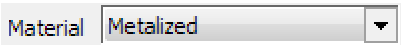
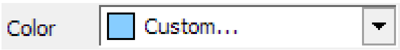
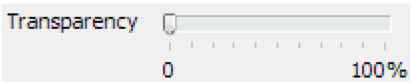
To apply a transformation to the current model, select a transformation type, set its parameters and click the *Apply* button (or hit Enter).

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).

Display properties

The elements selected in the model tree are highlighted in the 3D view. The following attributes can be defined below the model tree:

	Wireframe display
	Shaded surface display
	Shaded surface display with edges or isocurves
	Material
	Color ("Undefined" => default color of material)
	Transparency

The selection can be cleared by pressing the <Esc> key.

Model states

Model states contain the properties of all tree elements. Several model states can be managed via the controls above the model tree.

	Select existing model state
	Save model state
	Rename selected model state

+	Add new model state
−	Delete selected model state

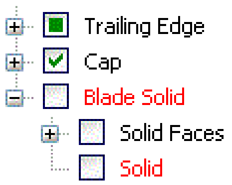
The following *predefined model states* cannot be modified:

- "Default" — The default model state
- "Default + CFD Setup" — The default model state with CFD Setup visible
- "Solids only" — Only solids are visible
- "Component colors" — Every component is displayed with the color defined in the Components view 168

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.

7.2.3 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 58.

Possible warnings

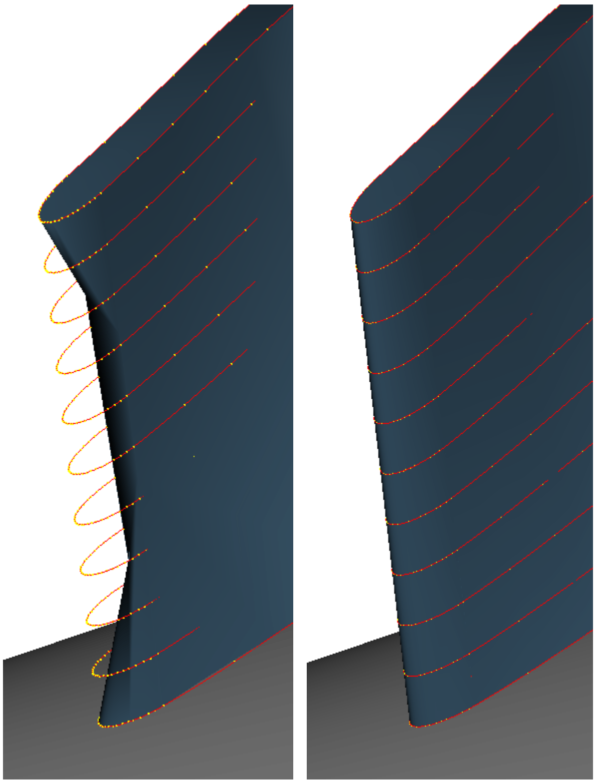
Problem	Possible solutions
3D-Error: Could not create solid ...	
Distance tolerance is too low or too high	Change the distance tolerance (see Model settings 376)

Problem	Possible solutions
Number of data points is disadvantageous (seldom)	Change the number of data points for the 3D model (see Model settings 376)

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- 376 or Volute-Settings 445)
- try a different display resolution (see Model display (top) 173)



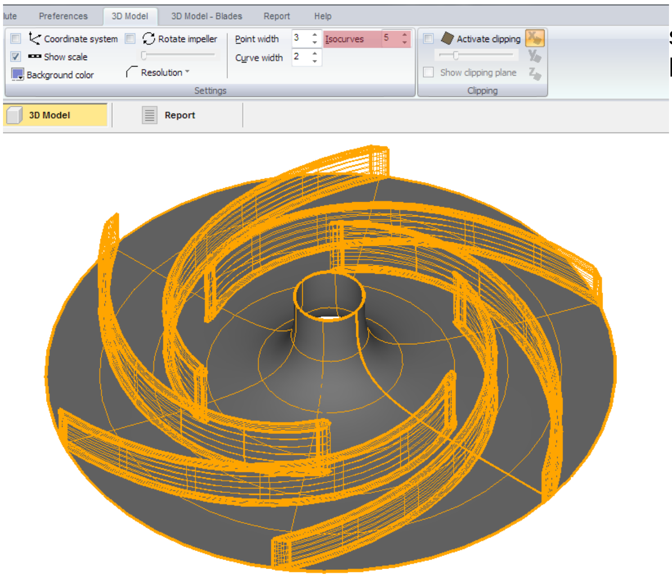
The pictures illustrate the possible influence of point density on the surface generation of the blade.

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) 173).

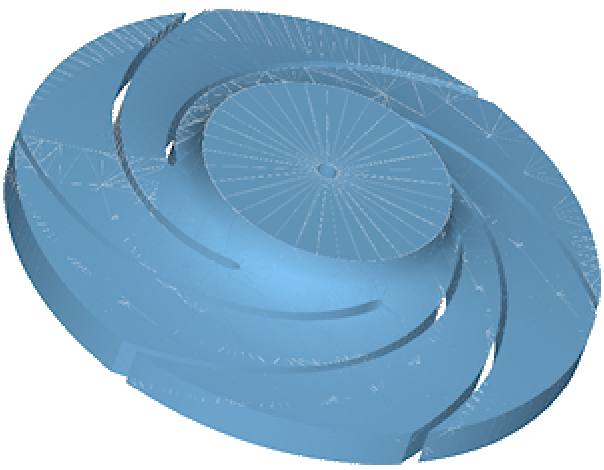
Normally, choosing another resolution (see Model display (top) 173) solves this problem.



The orange iso-curves show how the surface looks like.

Slow 3D model

If the handling of the 3D model is very slow, normally an update of the graphic card driver is helpful.



If problems occur in connection with the graphic card, sometimes an unsteady mesh is displayed on the faces of the solids.

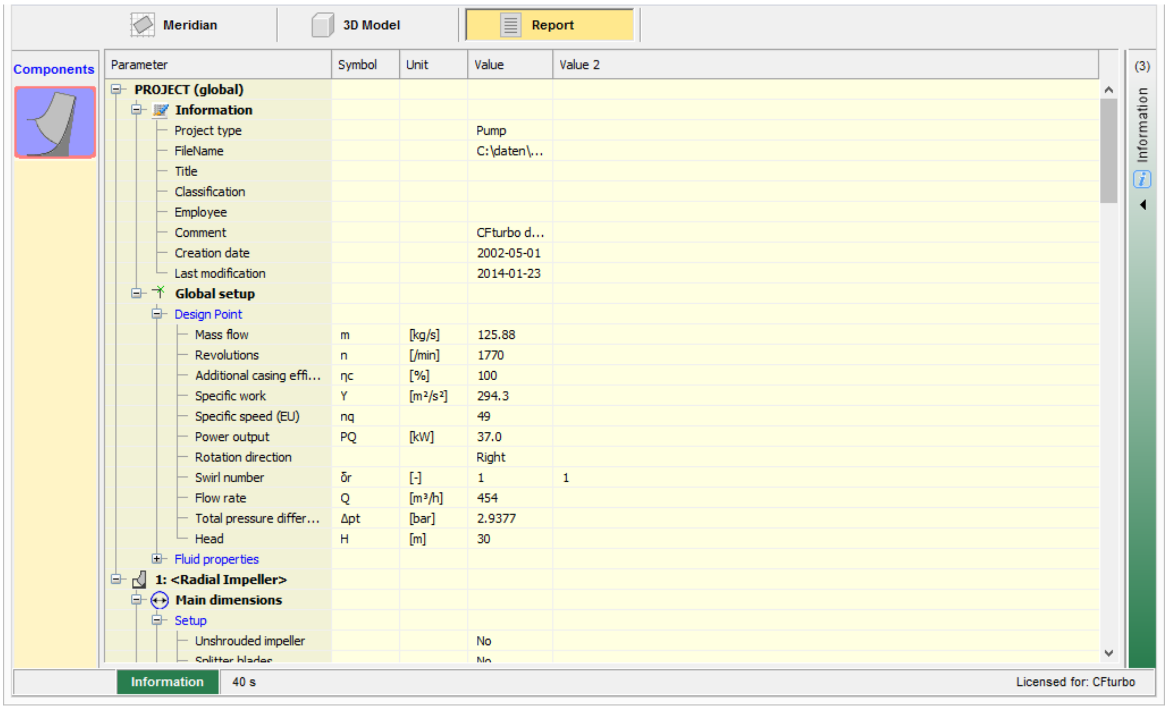
Visualization errors

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

7.3 Report

The report shows the most important information about the design in a tabular style.

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

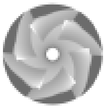
	Save report as HTML, RTF, CSV or TXT
	Print report
	Copy the content to the clipboard All marked rows are copied. If nothing is marked then all content is copied. Marking can be done by mouse, <Crtl> <A> marks all. Content will be pasted in MS Word/Excel as table.
	Expand all nodes

	Collapse all nodes

Part VIII

8 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 190
- Meridional contour 268
- Blade properties 292
- Blade mean lines 319
- Blade profiles 337
- Blade edges 344
- Model finishing 378
- Model settings 376
- CFD setup 368

Possible warnings

Problem	Possible solutions
The selected impeller shape (radial/ axial) is not matching with the specific speed.	
The impeller shape (radial/ mixed-flow or axial impeller) is not suitable for the selected design point 71. This warning is generated for	Select a suitable impeller shape corresponding to the specific speed calculated in the Global Setup 71: • radial/ mixed-flow impeller: nq ≈10 … 160

Problem	Possible solutions
• radial/ mixed-flow impellers with specific speed nq > 160 • axial impellers with specific speed nq < 80	• axial impeller: nq ≈ 80 … 400

8.1 Main dimensions

? Impeller \| Main dimensions

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

Details by impeller type

- Pump/Ventilator 191
- Compressor 227
- Turbine 240

Possible warnings

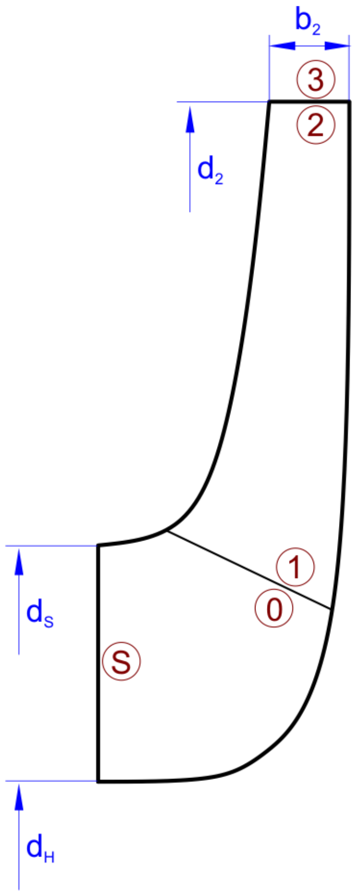
Problem	Possible solutions
Main dimensions are updated automatically. Therefore geometry modifications are possible.	
Main dimensions are updated automatically if any input parameters are modified.	To fix the main dimensions you could uncheck the "Automatic" calculation. Then you have to manually start the calculation if required.
Main dimensions are not updated automatically. Therefore the design could be not up-to-date.	
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.

Problem	Possible solutions
Hub inlet and outlet diameter seem to be in a wrong proportion.	
Potential min. hub outlet diameter (d_2-b_2) could be lower than inlet hub diameter d_H.	Increase impeller diameter d_2 or decrease impeller width b_2 or decrease hub diameter d_H.
Shroud inlet and outlet diameter seem to be in a wrong proportion.	
Potential max. shroud outlet diameter (d_2+b_2) could be lower than inlet shroud diameter d_S.	Increase impeller diameter d_2 or decrease impeller width b_2 or decrease shroud diameter d_S.
Specific speed of impeller is invalid.	
The specific speed nq of the impeller is much too low or too high.	Check design point 71 and power partitioning between impellers.
The selected impeller shape (radial/ axial) is not matching with the specific speed.	
The specific speed nq of the impeller is not suitable to the selected impeller type.	Select another impeller type (axial/ radial) or adapt the value for power partitioning between impellers.

8.1.1 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

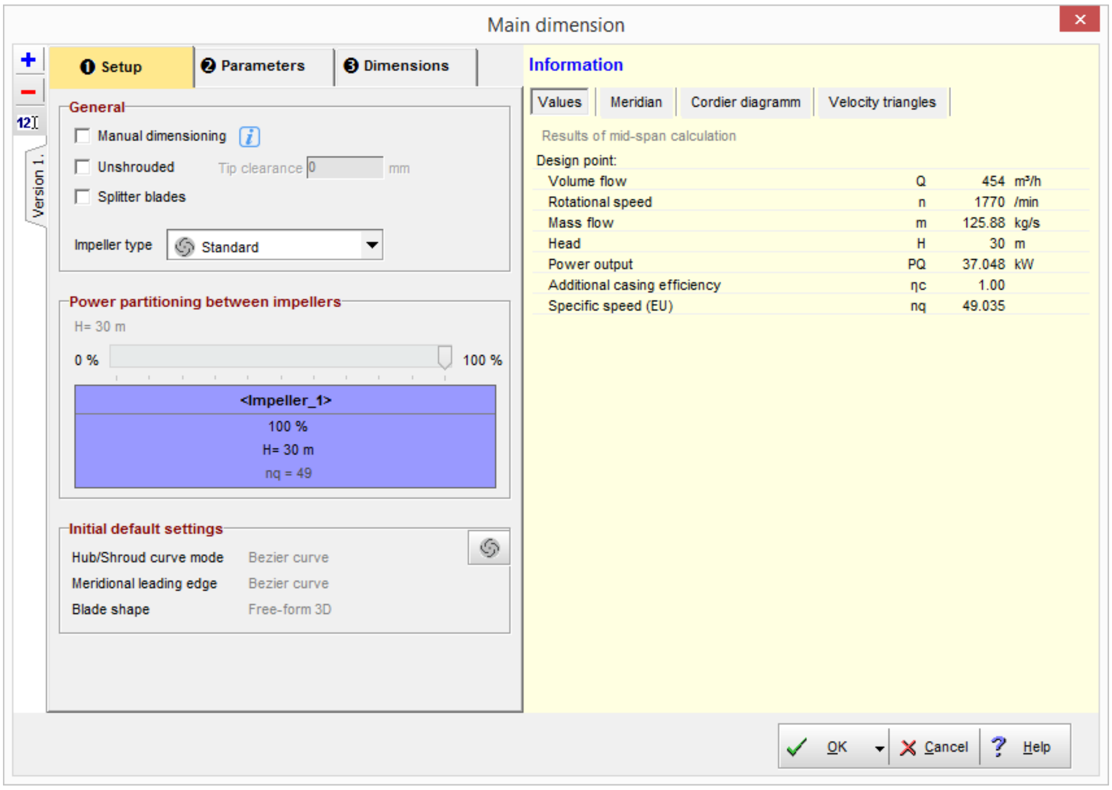
→ Setup 193

→ Parameters 194

→ Dimensions 201

8.1.1.1 Setup

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



On panel **General** you can select:

- **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.

- **Splitter blades (not for axial ventilators)**
  Design impeller with or without splitter blades.

- **Unshrouded**
  Design a shrouded (closed) or unshrouded (open) impeller.
  For an unshrouded impeller you have to define the **tip clearance**.

- **Impeller type**
  For pumps select between **Standard** impeller and **Wastewater** impeller type. For wastewater pump impellers you have to specify the desired number of blades used for some specific empirical correlations.

In case more than 1 impeller is contained in the project the design point 71 (head, pressure difference etc.) can be distributed amongst the impellers using the power partitioning. The energy goal used for the design of the selected impeller (index i) is determined by:

$$E_i = e_i \cdot E_{Global} \,,$$

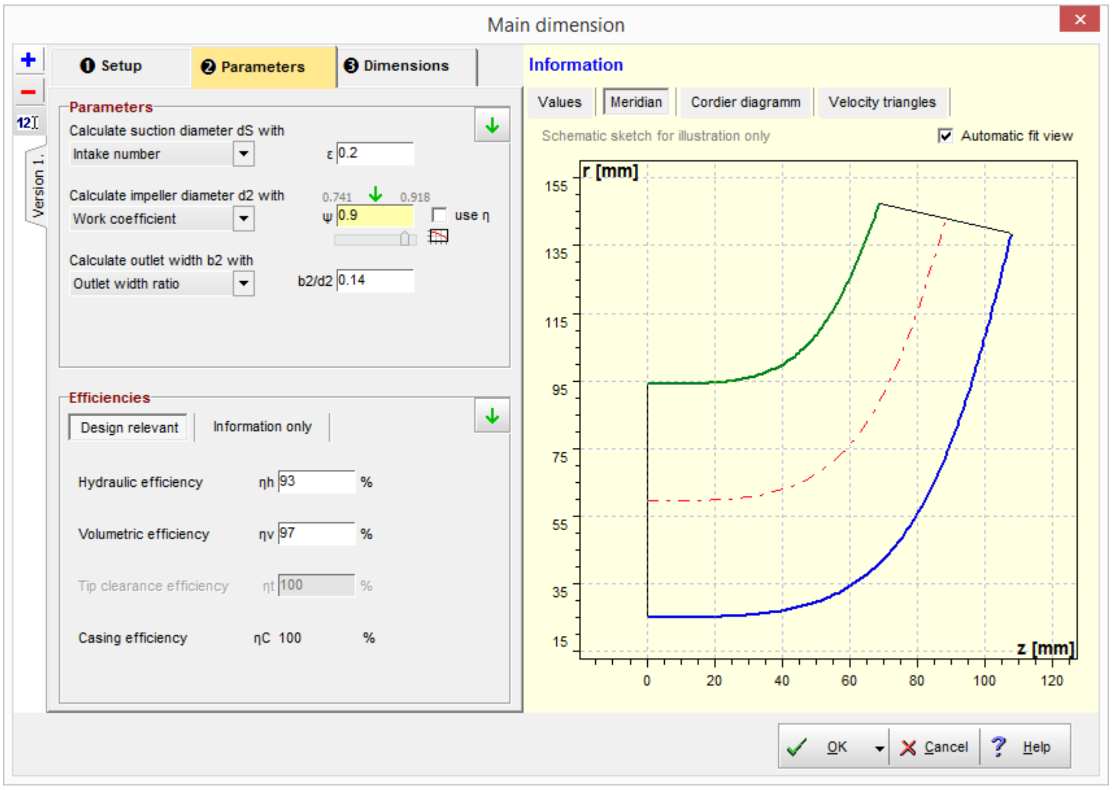
where the capital E may either be head, specific work or pressure difference resp. The lower case e_i is the ratio describing the power partitioning for the selected impeller.

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 161 for more information.

Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see Global setup 71).

8.1.1.2 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 145.

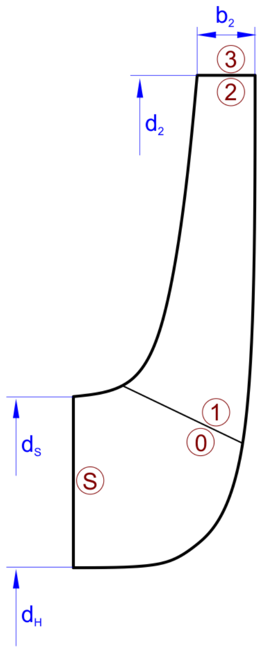


For details of how to handle the parameter edit fields please see Edit fields with empirical functions 47.

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 ε	▪ 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}	▪ high → smaller dimensions, lower friction losses ▪ < 20° → prevent the risk of cavitation ▪ > 15° → with regard to efficiency ▪ 12°…17° → recommended for good suction capability
Minimal relative	▪ small friction and shock losses

velocity w	▪ only if no cavitation risk ! ▪ f_{dS}=1.15...1.05 standard impeller, nq=15...40 ▪ f_{dS}=1.25...1.15 suction impeller
suction specific speed n_{SS}	$n_{SS} = n\left[min^{-1}\right] \cdot \frac{\sqrt{Q\left[m^3/s\right]}}{\left(NPSH_R\ [m]\right)^{3/4}}$ (European definition for illustration) Standard suction impeller u_1<50 m/s 160...220 Suction impeller, axial inflow u_1<35 m/s 220...280 Suction impeller, cont. shaft u_1<50 m/s 180...240 High pressure pump u_1>50 m/s 160...190 Standard inducer u_1>35 m/s 400...700 Rocket inducer >>1000
Min. NPSH	$NPSH_R = \lambda_c \frac{c_{m1}^{\ 2}}{2g} + \lambda_w \frac{w_1^{\ 2}}{2g}$ ▪ 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

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}	

For d_2-calculation

Work coefficient ψ	▪ dimensionless expression for the specific energy: $\psi = Y/\left(u_2^{\,2}/2\right)$ and $\psi = Y_{eff}/\left(u_2^{\,2}/2\right)$ 0.7 ...1.3 radial impeller 0.25...0.7 mixed-flow impeller 0.1 ...0.4 axial impeller ▪ high → small d_2, flat characteristic curve low → high d_2, steep characteristic curve ▪ If the check box "**use η**" is set d_2-calculation is done on the basis of Y_{eff}=Y/η. Otherwise Y - specific work without losses - is used.
Diameter coefficient δ	▪ according to Cordier diagram (see Dimensions 201)
Outflow angle β_3	▪ 6°...13°: recommended for stable performance curve (with nq rising)

For b_2-calculation

Outlet width ratio b_2/d_2	▪ 0.04...0.30 (rising with nq)
for pumps: Mer. deceleration c_{m3}/c_{mS}	▪ 0.60...0.95 (rising with nq)
for pumps: Outlet coefficient ε_2	▪ Ratio between meridional outlet velocity and specific energy $\varepsilon_2 = c_{m2}/\sqrt{2Y}$ ▪ 0.08...0.26 (rising with nq) ▪ (k_{m2} at Stepanoff)
for ventilators: Shroud angle ε_{Shr}	

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

Information only

- side friction efficiency η_S
- mechanical efficiency η_m
- motor efficiency η_{mot}
- casing efficiency η_c (displayed for information only, see Global setup[71])

The casing efficiency η_c is used additionally for impeller dimensioning in order to compensate the flow losses in the casing.

The losses resulting in energy dissipation from the fluid form the **impeller efficiency**.

$$\eta_{Im} = \eta_h \eta_v \eta_S \eta_T$$

Impeller, casing 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.

P_Q: pump output, see above

P_D: mechanical power demand (coupling/ driving power)

P_{el}: electrical power demand of motor

The following summary illustrates the single efficiencies and their classification:

classification		**efficiencies**		**Relevant for impeller design**
stage	casing	η_C	casing	**yes**: for energy transmission
	impeller	η_h	hydraulic	
		η_T	tip	
		η_V	volumetric	**yes**: for flow rate
		η_S	side friction	**no**: for overall information only
	mechanical	η_m	mechanical	
stage incl. motor	electrical	η_{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 145 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:

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

(rising with impeller size)

The tip clearance efficiency is only relevant for unshrouded impellers. It contains losses due to the flow through the gap between blade tips and housing from the pressure to the suction side of the 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.

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

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

$$\eta_S = 1 - \frac{P_S}{P} \approx \begin{matrix} 0.5 \ldots 0.985 & \text{für } n_q < 40 \\ 0.985 \ldots 0.995 & \text{für } n_q > 40 \end{matrix}$$

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

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

(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.

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	$P_L = P_D - P_Q = P_D(1 - \eta_{St})$
Impeller efficiency	$\eta_{Im} = \eta_h \eta_v \eta_S \eta_T$
Stage efficiency	
Stage efficiency incl. motor	

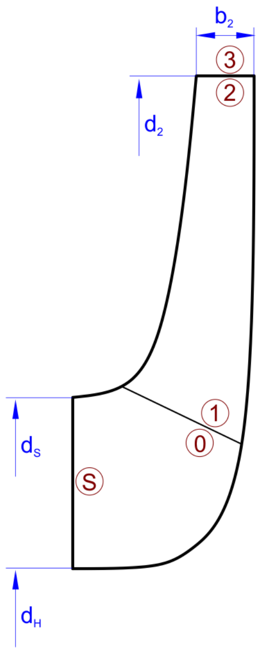
8.1.1.3 Dimensions

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

→ Shaft/Hub 267

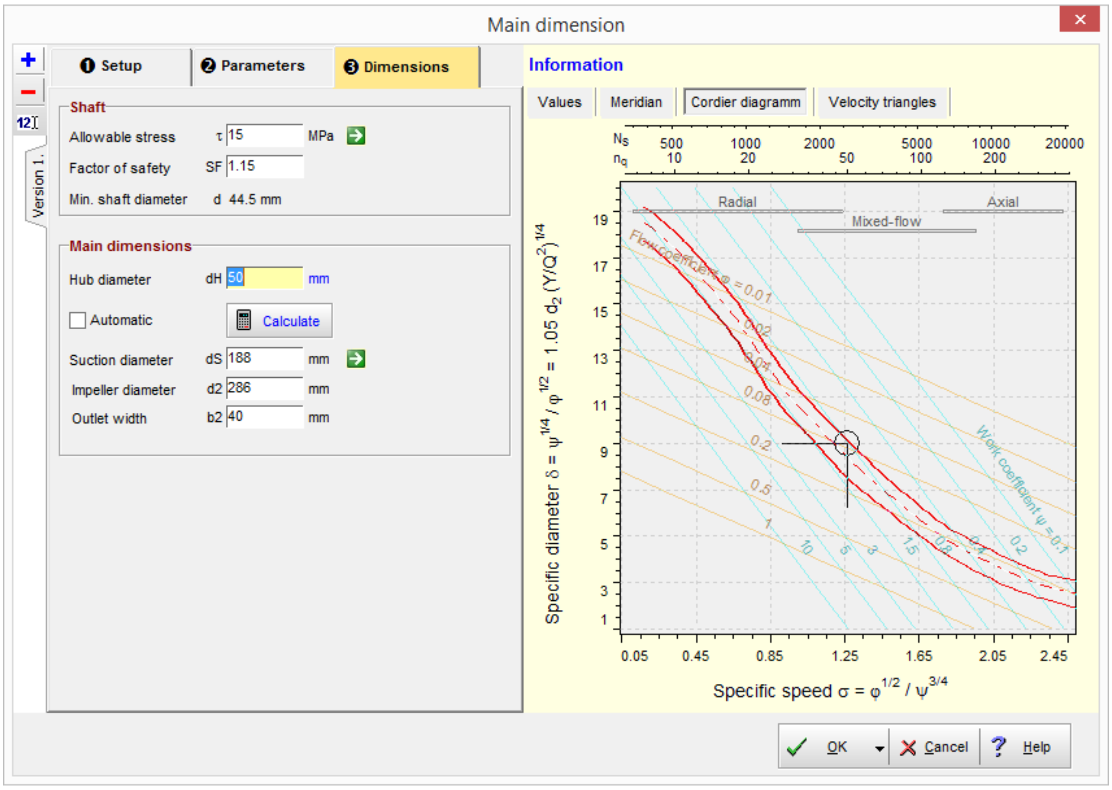
The main dimensions of a designed impeller - suction diameter d_S, impeller diameter d_2, outlet width b_2 - can be seen on **Main dimensions** panel. They can be recomputed by pressing the **Calculate**-button. The computation is based on "Euler's Equation of Turbomachinery", on the continuity equation and the relations for the velocity triangles as well as on the parameters and parameter ratios given in the tab sheets **Setup** and **Parameters**.

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

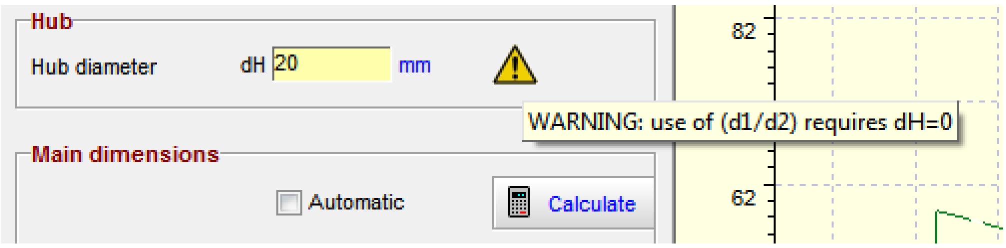


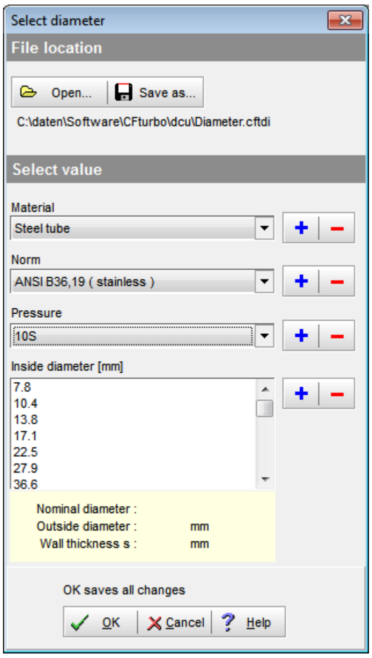
In case the checkbox **Automatic** is activated a new calculation will accomplished after any change of parameter. Then the manual alteration of the main dimensions is not possible.

Regarding the impeller size one should try to attain d_2 values as low as possible. But there is a limit for a specified task: lower impeller diameters are leading to higher blade loading - up to blade angles β_2 which may not be suitable anymore.



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_N = 0. Otherwise the empirical correlations are invalid. If the user defines a d_N 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 194).





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.cftdi** and is located in the installation directory of CFturbo. Modifications of the list will be saved if the user is leaving the dialog window by clicking the **OK**-button. In case there are no write permissions the user can choose another directory to save the file. Renaming of files is possible by **Save as**- functionality. By clicking the **Open**-button a previously saved file can be opened.

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 205 as well as the Cordier-Diagramm 205 with the location of the best point. These three sections can be chosen by the appropriate soft buttons in the heading.

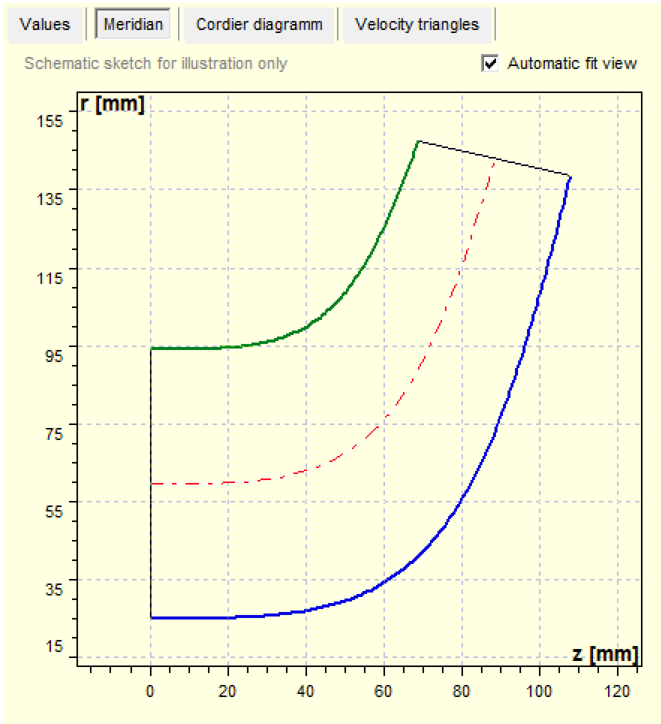
In the **Value** section the following variables are displayed for information which result from calculated or determined main dimensions:

Work coefficient	$\psi = \dfrac{Y}{u_2^2/2}$
Flow coefficient	
Meridional flow coefficient	

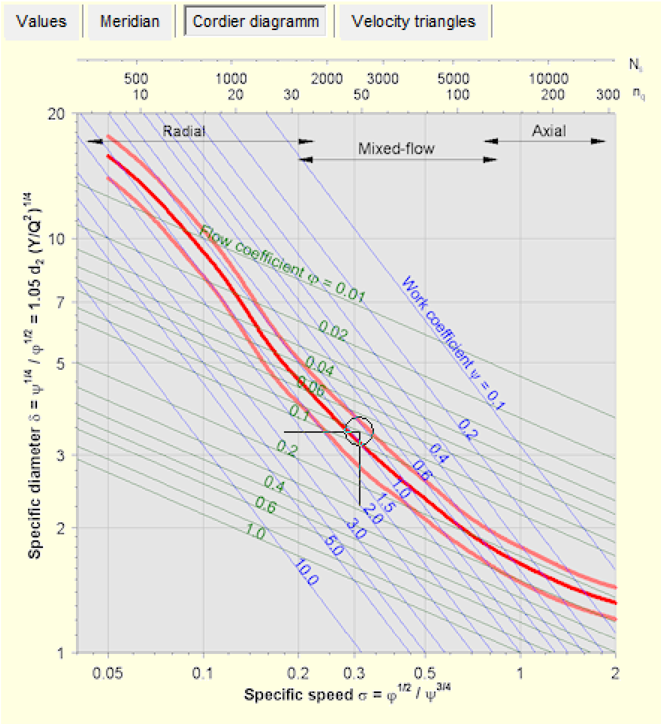
Diameter coefficient	$\delta = \frac{\psi^{1/4}}{\varphi_t^{1/2}} = 1.05 d_2 \left(\frac{Y}{Q^2}\right)^{1/4}$
Average inlet velocity	$\bar{c}_{mS} = \frac{Q/\eta_v}{\pi/4\left(d_S^2 - d_N^2\right)}$
Average inlet velocity (net)	$\bar{c}_{mS}^* = \frac{Q}{\pi/4\left(d_S^2 - d_N^2\right)}$
Average outlet 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	Pfleiderer $NPSH_R = \lambda_c \frac{c_{m1}^2}{2g} + \lambda_w \frac{w_1^2}{2g}$ with loss coefficients λ_c = 1.1 … 1.35, λ_w = (0.03) 0.1 … 0.3
	Gülich $NPSH_R = H \cdot \left(n_q / n_{SS}\right)^{4/3}$ or $NPSH_R = \left(n\sqrt{Q} / n_{SS}\right)^{4/3}$ with suction specific speed n_{SS} = 160…280
	Stepanoff $NPSH_R = \sigma \cdot H$ with cavitation number σ = 1.22 $\cdot 10^{-3} \cdot n_q^{4/3}$
	Petermann with suction number S_q = (0.2) 0.4…0.6 (2.0)
	Europump

Outlet width ratio	b_2/d_2
Meridional deceleration	$d_{cm} = \bar{c}_{m3} / \bar{c}_{mS}$
Estimated axial force	$F_{ax} = 0.9 \rho g H \cdot \pi / 4 \left(d_S{}^2 - d_N{}^2 \right)$

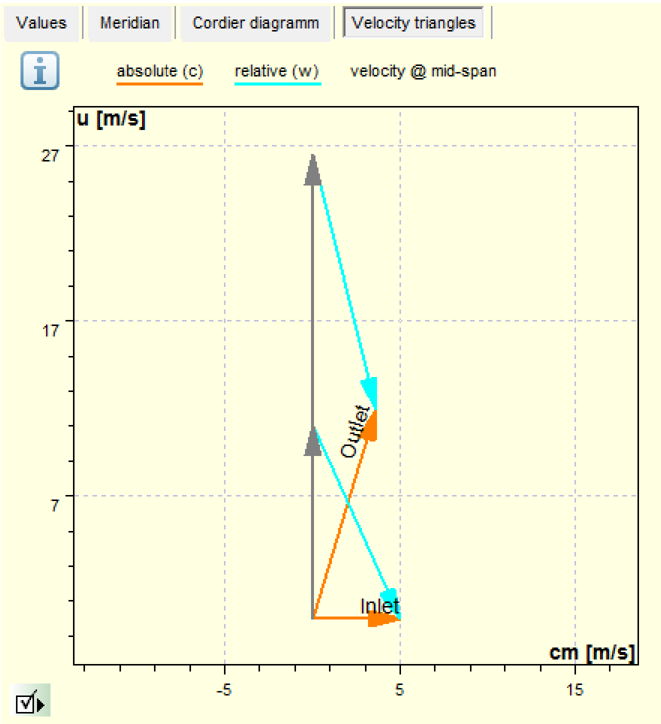
The **Meridional preview** is until now based on the main dimensions only.



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



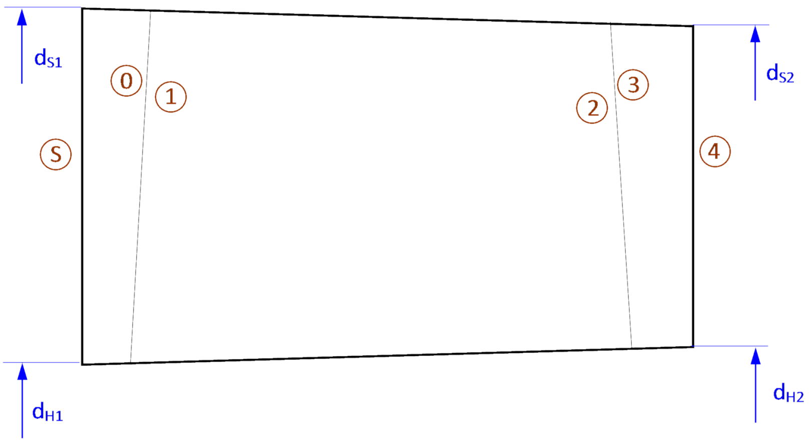
The **Velocity triangles** are the result of a mid-span calculation and are based on the design point 71 and the main dimensions.



8.1.2 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 208

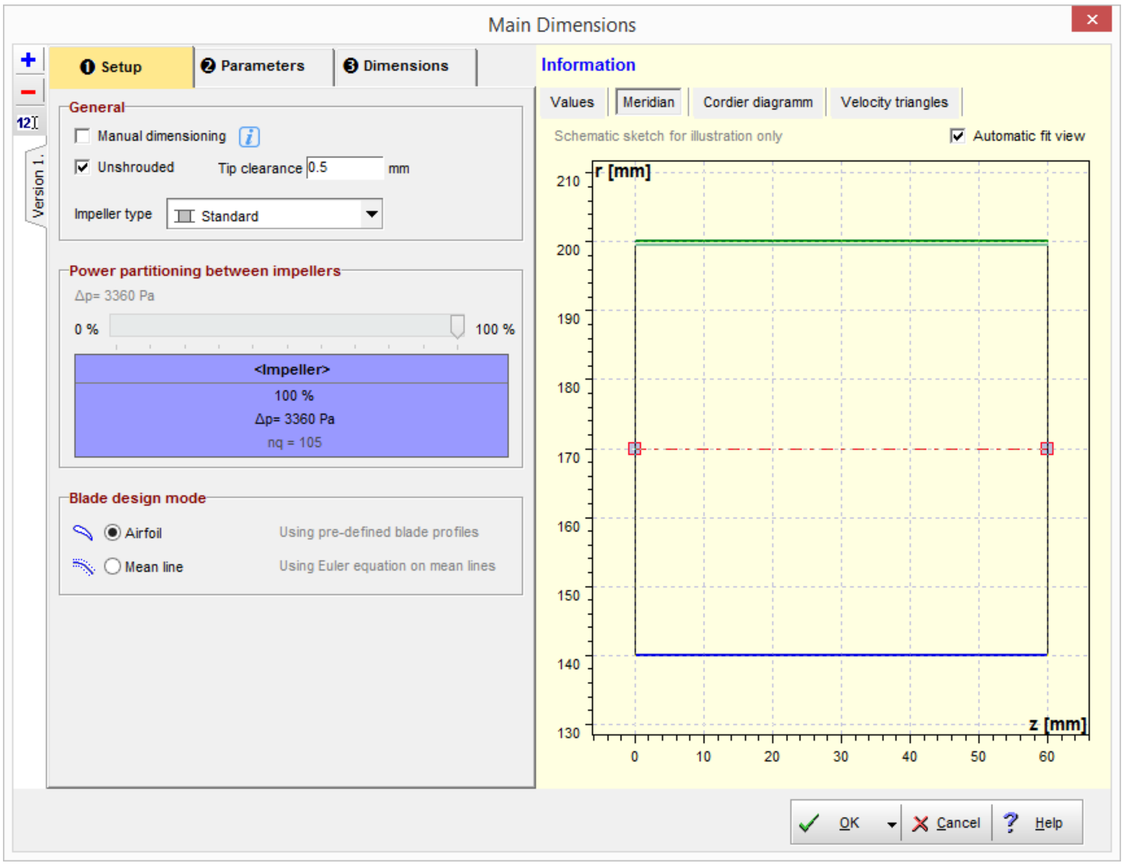
→ Pump: Parameters 210

→ Ventilator: Parameters 217

→ Dimensions 221

8.1.2.1 Setup

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



General

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

- **Unshrouded**
  Design a shrouded (closed) or unshrouded (open) impeller.
  For an unshrouded impeller you have to define the **tip clearance**.

- **Impeller type**
  For pumps select between **Standard** impeller and **Inducer** impeller type.
  For Ventilators select between **Standard** impeller and **Automotive cooling** impeller type.

Power partitioning between impellers

In case more than 1 impeller is contained in the project the design point 71 (head, pressure difference etc.) can be distributed amongst the impellers using the power partitioning. The energy goal used for the design of the selected impeller (index i) is determined by:

$$E_i = e_i \cdot E_{Global} \text{ ,}$$

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 partitioning for the selected impeller.

Blade design mode

- **Airfoil/Hydrofoil** 351
  Design according to Airfoil/Hydrofoil design theory.
- **Mean line** 292
  Design using Euler's equation on mean lines.

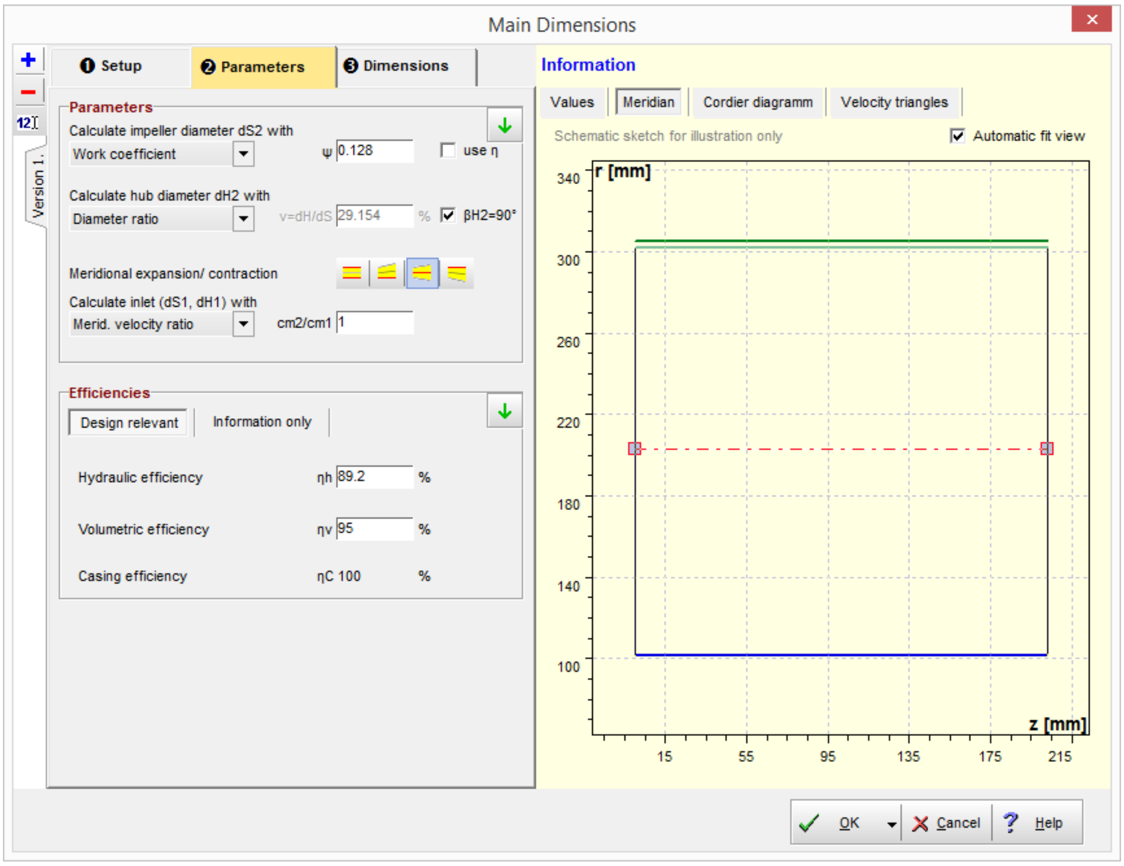
Initial default setting

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 161 for more information.

Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see Global setup 71).

8.1.2.2 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 145.

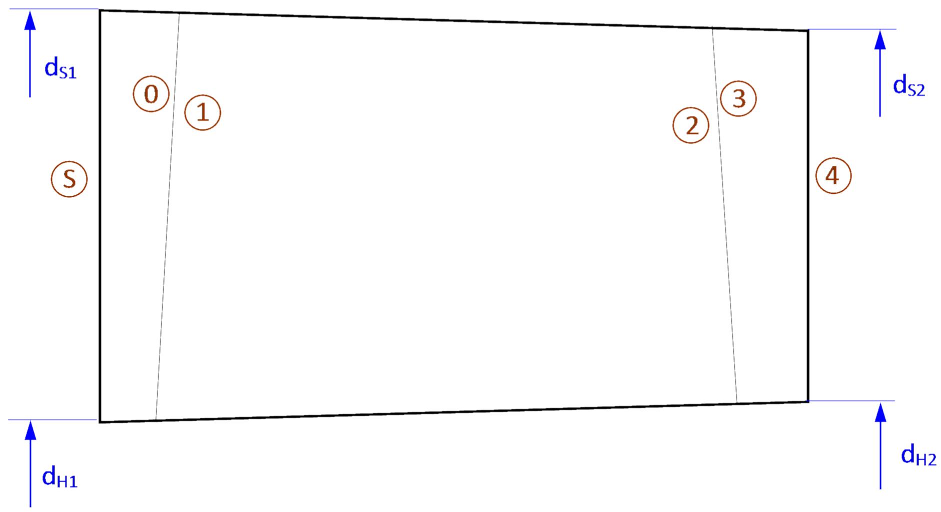


For details of how to handle the parameter edit fields please see Edit fields with empirical functions 47.

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 215 special correlations are used.

For d_{S2}-calculation

Work coefficient ψ	▪ dimensionless expression for the specific energy: $\psi = Y/(u_2{}^2/2)$ and $\psi = Y_{eff}/(u_2{}^2/2)$ 0.7 ...1.3 radial impeller 0.25...0.7 mixed-flow impeller 0.1 ...0.6 axial impeller ▪ high → small d_{S2}, flat characteristic curve low → high d_{S2}, steep characteristic curve ▪ If the check box "**use η**" is set d_{S2}-calculation is done on the basis of Y_{eff}=Y/η. Otherwise Y - specific work without losses - is used.
Diameter coefficient δ	▪ according to Cordier diagram (see Dimensions 221)

For d_{H2} calculation

Diameter ratio d_{H2}/d_{S2}	$\frac{d_{H2}}{d_{S2}} = 0.4 \ldots 0.9$ If the check box "**β_{H2} = 90°**" is set the diameter ratio is set to: $\frac{d_{H2}}{d_{S2}} = \frac{\sqrt{Y}}{u_{S2}}$ Under the assumptions: $c_u \cdot u = Y = const.$

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

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

Efficiency

In panel **Efficiency** you have to specify several efficiencies. You have to distinguish between design relevant efficiencies and efficiencies used for information only:

Design relevant

- hydraulic efficiency η_h
- volumetric efficiency η_v

Information only

- mechanical efficiency η_m
- motor efficiency η_{mot}

The casing efficiency η_c is used additionally for impeller dimensioning in order to compensate the flow losses in the casing.

The losses resulting in energy dissipation from the fluid form the **impeller efficiency**.

$$\eta_{Im} = \eta_h \cdot \eta_v$$

Impeller, casing 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.

$$\eta_{St} = \frac{P_Q}{P_D} = \eta_{Im}\eta_c\eta_m$$

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:

<table>
<tr><th colspan="2">classification</th><th colspan="2">efficiencies</th><th>Relevant for impeller design</th></tr>
<tr><td rowspan="4">stage</td><td>casing</td><td>η_C</td><td>casing</td><td rowspan="2">**yes**: for energy transmission</td></tr>
<tr><td rowspan="2">impeller</td><td>η_h</td><td>hydraulic</td></tr>
<tr><td>η_V</td><td>volumetric</td><td>**yes**: for flow rate</td></tr>
<tr><td>mechanical</td><td>η_m</td><td>mechanical</td><td rowspan="2">**no**: for overall information only</td></tr>
<tr><td>stage incl. motor</td><td>electrical</td><td>η_{mot}</td><td>motor</td></tr>
</table>

The obtainable overall efficiency correlates to specific speed and to the size and the type of the impeller as well as to special design features like bypass installations and auxiliary aggregates. Efficiencies calculated by approximation functions 145 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:

$$\eta_V = \frac{Q}{\tilde{Q}} \approx 0.70 \ldots 0.95$$

(rising with decreasing tip clearance)

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

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

(rising with impeller size)

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

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	$P_L = P_D - P_Q = P_D(1 - \eta_{St})$
Impeller efficiency	$\eta_{Im} = \eta_h \cdot \eta_v$
Stage efficiency	$\eta_{St} = \frac{P_Q}{P_D} = \eta_{Im}\eta_m\eta_c$
Stage efficiency incl. motor	

8.1.2.2.1 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:

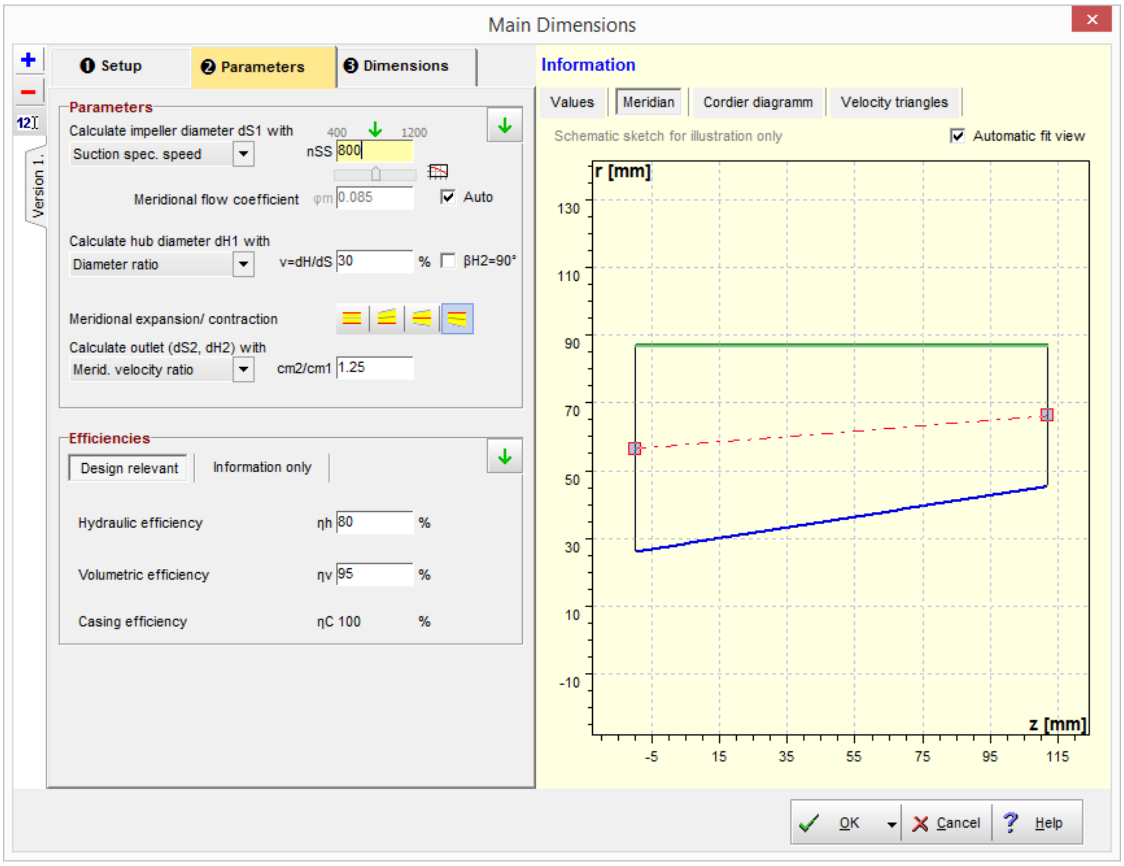
$$\varphi_m = \frac{Q}{A_S u_{S1}} = \frac{4Q}{\pi\left(d_{S1}^2 - d_{H1}^2\right)\cdot \pi d_{S1} n} = \frac{c_{m1}}{u_{S1}} = \tan\beta_{0S}$$

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}:

$$n_{SS} = n\left[\min^{-1}\right]\cdot\frac{\sqrt{Q\left[m^3/s\right]}}{\left(NPSH_R\ [m]\right)^{3/4}}$$

(or the US definition N_{ss}, see Preferences/Units/Other 160)

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

Alternatively you can specify the rel. inlet flow angle β_{0S} or the meridional flow coefficient φ_m directly. Furthermore the parameters for classic axial pump 210 design could be used alternatively.

The inlet hub diameter d_{H1} is calculated using the diameter ratio ν_1:

$$\nu_1 = \frac{d_{H1}}{d_{S1}} \approx 0.2 \ldots 0.4$$

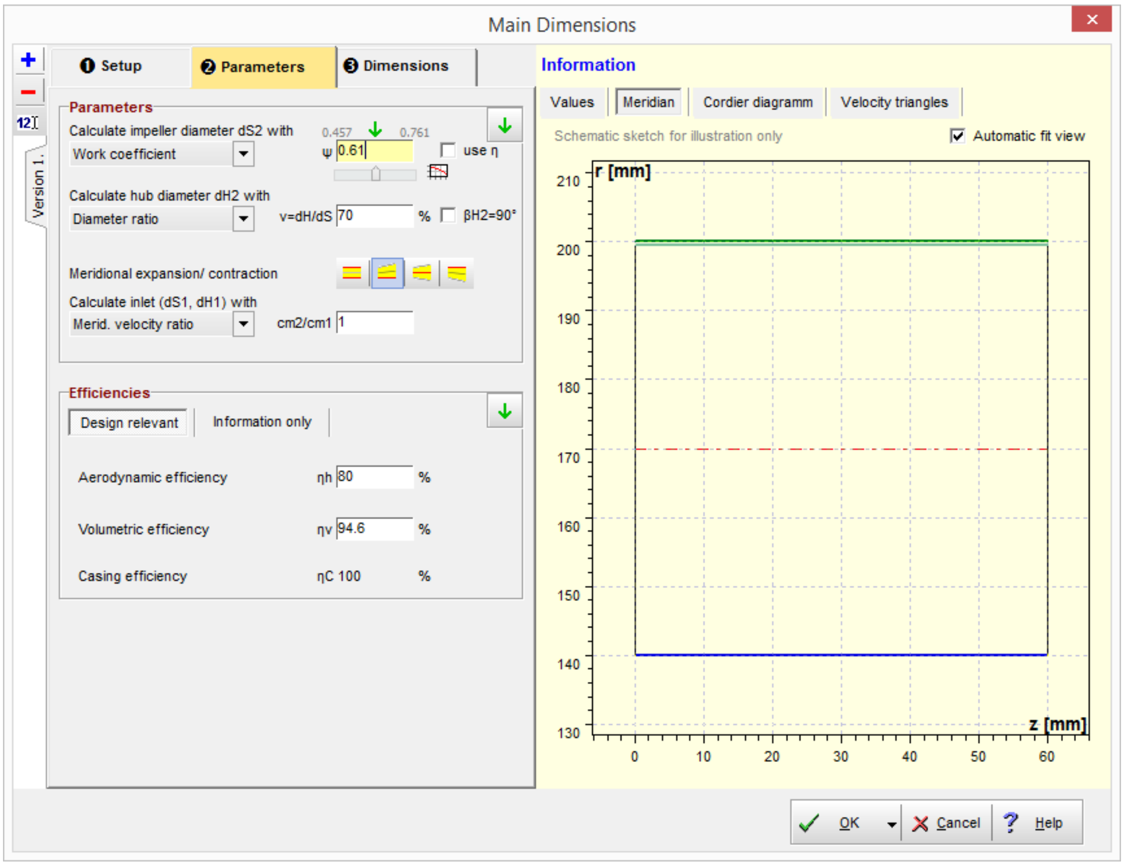
Typical for inducers is a constant tip (shroud) diameter. The hub diameter can increase from inlet to outlet slightly in order to use centrifugal effect for energy transmission. The meridional velocity ratio between inlet and outlet can be used to estimate the outlet cross section:

$$\frac{c_{m2}}{c_{m1}} \approx 1 \ldots 1.5$$

Alternatively the diameter ratio $\nu_2 = d_{H2}/d_{S2}$ at outlet similar to the inlet side can be used.

8.1.2.3 Parameters Ventilator

On page **Parameters** you have to put in or to modify parameters resulting from approximation functions in dependence on specific speed nq or flow rate Q. See Approximation functions 145.

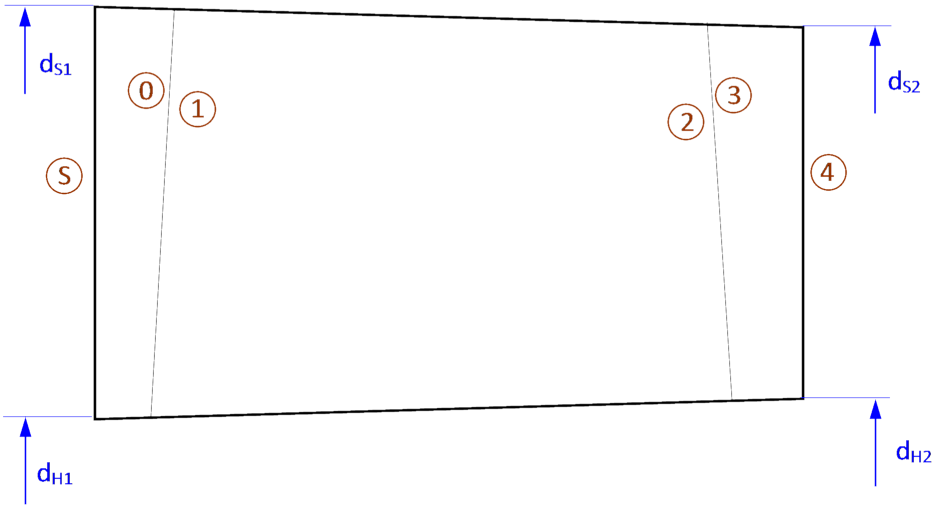


For details of how to handle the parameter edit fields please see Edit fields with empirical functions 47.

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

Work coefficient ψ	▪ dimensionless expression for the specific energy: $\psi = Y/\left(u_2^{\ 2}/2\right)$ and $\psi = Y_{eff}/\left(u_2^{\ 2}/2\right)$ 0.7 ...1.3 radial impeller 0.25...0.7 mixed-flow impeller 0.1 ...0.6 axial impeller ▪ high → small d_{S2}, flat characteristic curve low → high d_{S2}, steep characteristic curve ▪ If the check box "**use η**" is set d_{S2}-calculation is done on the basis of Y_{eff}=Y/η. Otherwise Y - specific work without losses - is used.

For d_{H2} calculation

Diameter ratio d_{H2}/d_{S2}	If the check box "**β_{H2} = 90°**" is set the diameter ratio is set to: Under the assumptions: $c_u \cdot u$ = Y = const.

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

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

Efficiency

In panel **Efficiency** you have to specify several efficiencies. You have to distinguish between design relevant efficiencies and efficiencies used for information only:

Design relevant

- Total-total efficiency η_{tt}
- volumetric efficiency η_v

Information only

- mechanical efficiency η_m
- motor efficiency η_{mot}

The casing efficiency η_c is used additionally for impeller dimensioning in order to compensate the flow losses in the casing.

The losses resulting in energy dissipation from the fluid form the **impeller efficiency**.

Impeller, casing 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.

$$\eta_{St} = \frac{P_Q}{P_D} = \eta_{Im}\eta_c\eta_m$$

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:

<table>
<tr><th colspan="2">classification</th><th colspan="2">efficiencies</th><th>Relevant for impeller design</th></tr>
<tr><td rowspan="4">stage</td><td>casing</td><td>η_C</td><td>casing</td><td rowspan="2">yes: for energy transmission</td></tr>
<tr><td rowspan="2">impeller</td><td>η_{tt}</td><td>total-total</td></tr>
<tr><td>η_V</td><td>volumetric</td><td>yes: for flow rate</td></tr>
<tr><td>mechanical</td><td>η_m</td><td>mechanical</td><td rowspan="2">no: for overall information only</td></tr>
<tr><td>stage incl. motor</td><td>electrical</td><td>η_{mot}</td><td>motor</td></tr>
</table>

The obtainable overall efficiency correlates to specific speed and to the size and the type of the impeller as well as to special design features like bypass installations and auxiliary aggregates. Efficiencies calculated by approximation functions 145 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 which also includes the circulating flow within the ventilator:

(rising with decreasing tip clearance)

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

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

(rising with impeller size)

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

Information

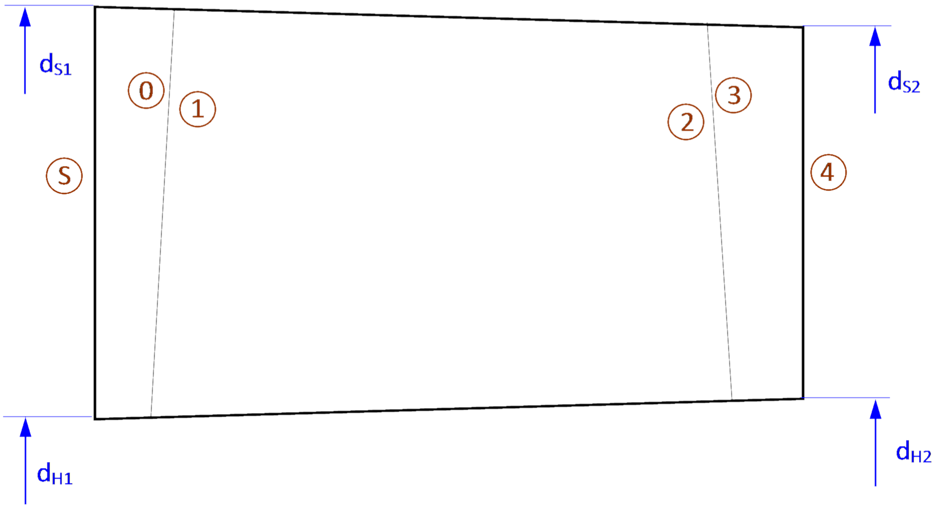
In the right area of the register **Parameter** you can find again some calculated values for **information**:

Required driving power	$P_D = \frac{P_Q}{\eta_{St}}$
Power loss	$P_L = P_D - P_Q = P_D(1 - \eta_{St})$
Impeller efficiency	$\eta_{Im} = \eta_{tt}\eta_V$
Stage efficiency	$\eta_{St} = \frac{P_Q}{P_D} = \eta_{Im}\eta_m\eta_c$
Stage efficiency incl. motor	$\eta_{St}^* = \frac{P_Q}{P_{el}} = \eta_{St}\eta_{mot}$

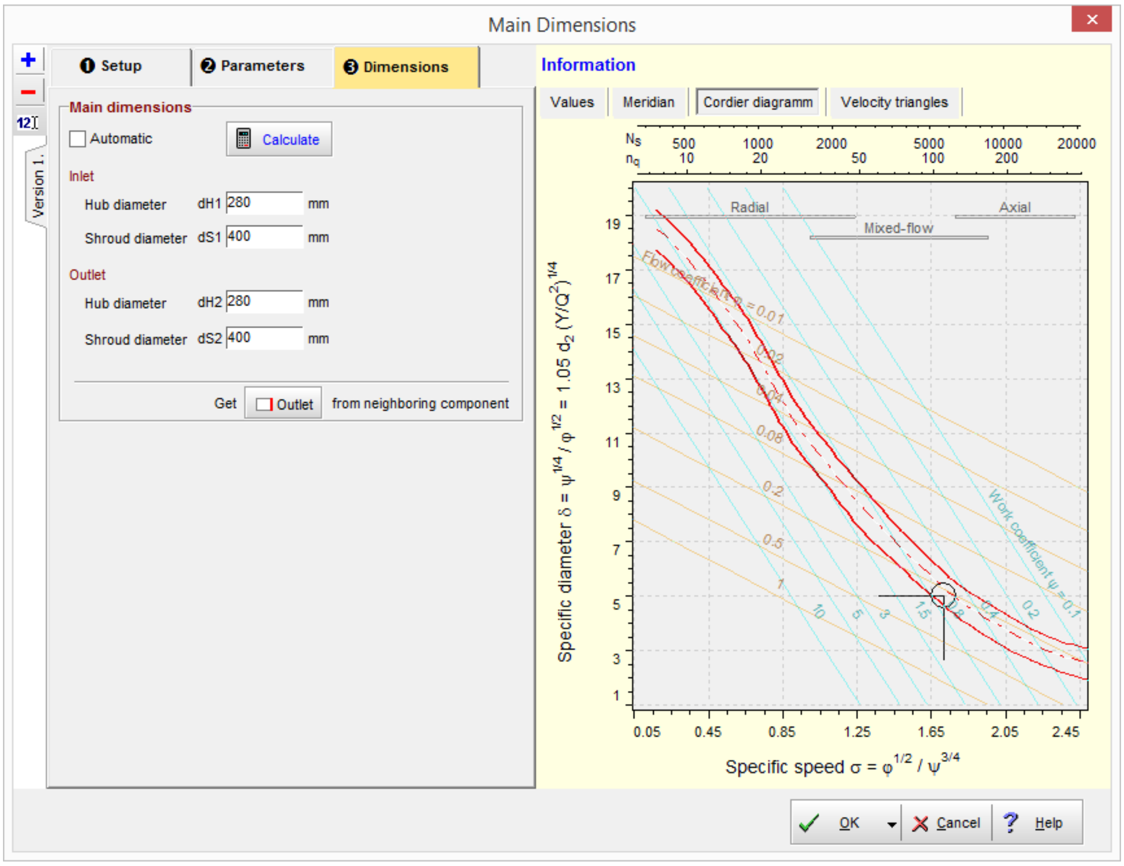
8.1.2.4 Dimensions

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**.

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



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



Information

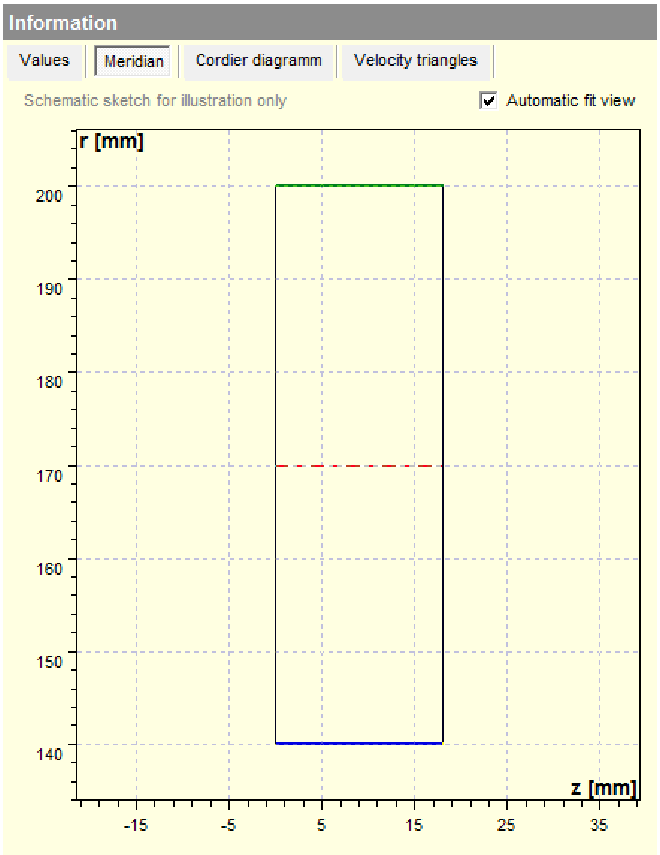
In the right panel of any tab sheet an **information** panel is situated, which holds the computed variables in accordance to the actual state of design, the resulting Meridional section 205 as well as the Cordier-Diagramm 205 with the location of the best point. These three sections can be chosen by 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:

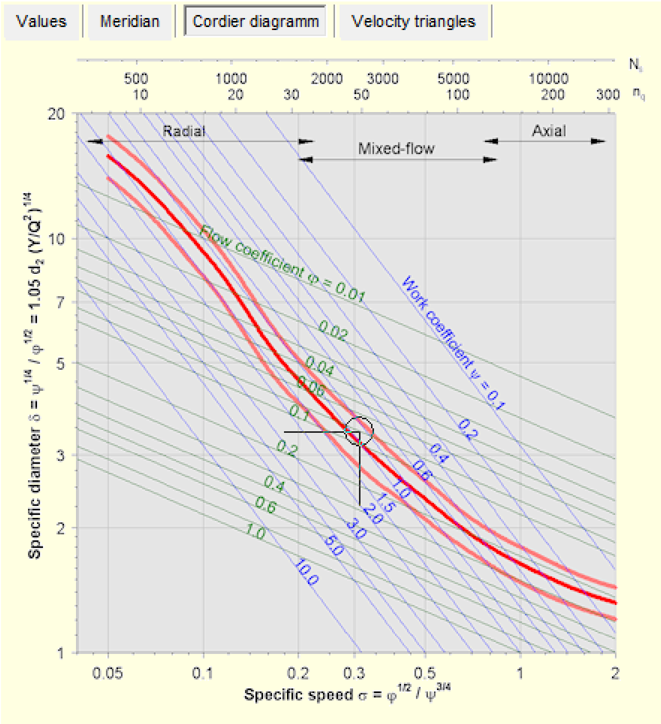
Work coefficient	
Flow coefficient	

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

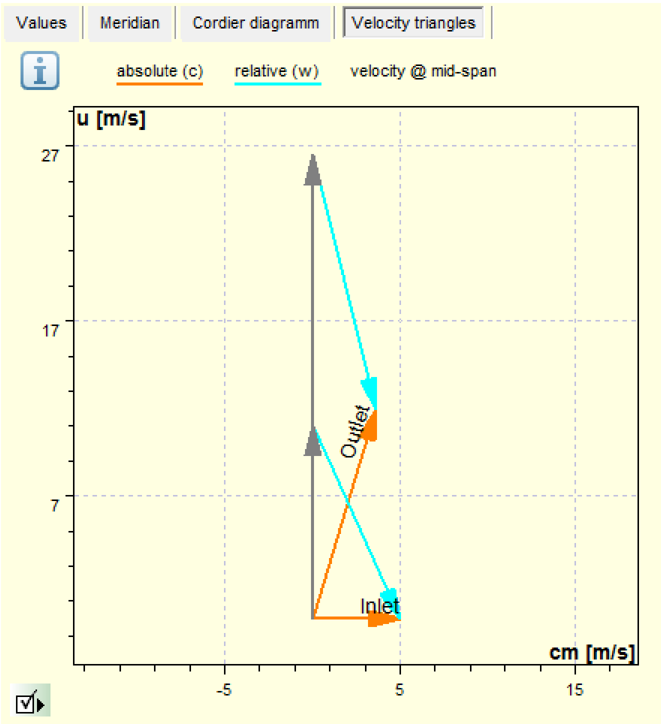
The **Meridional preview** is until now based on the main dimensions only.



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



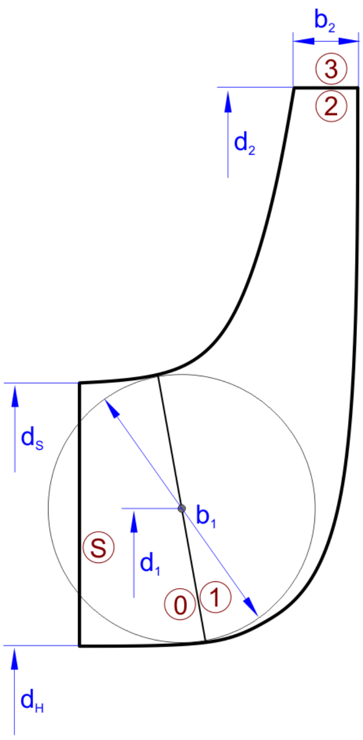
The **Velocity triangles** are the result of a mid-span calculation and are based on the [design point](71) and the main dimensions.



8.1.3 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

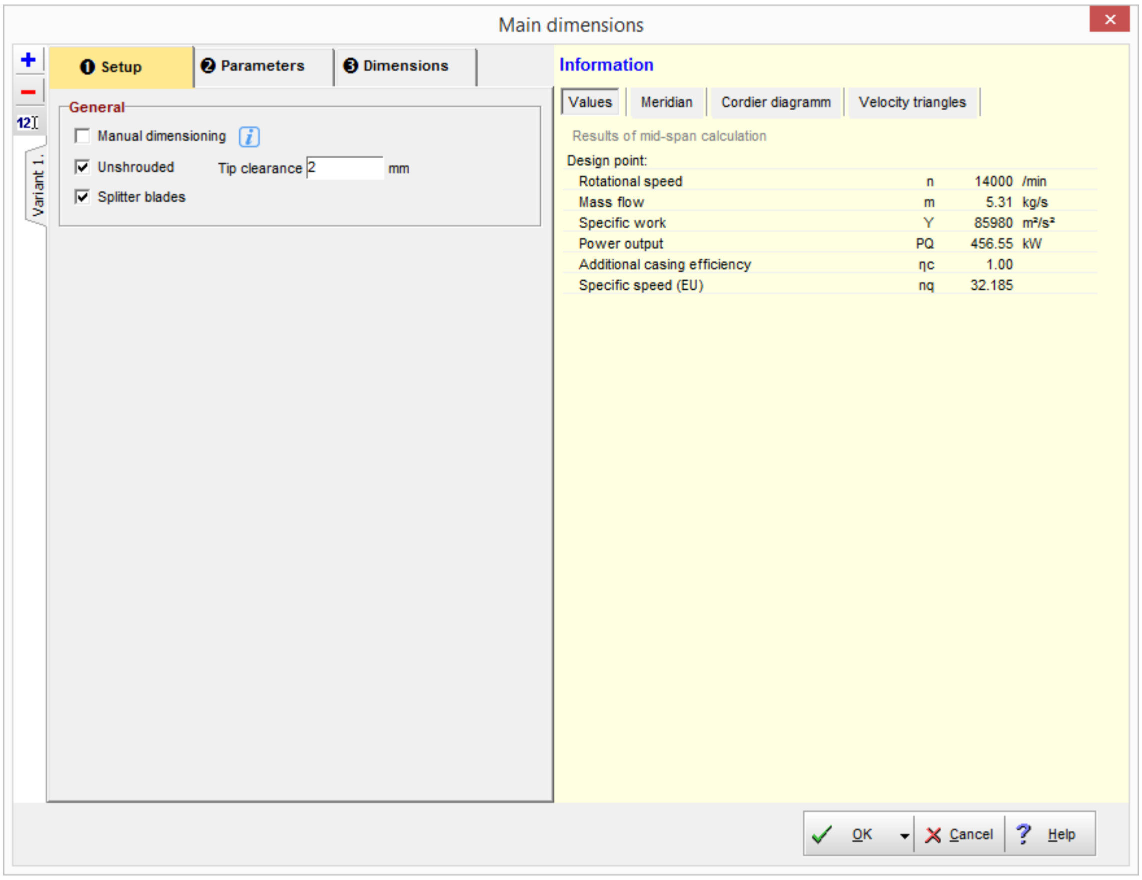
→ Setup 228

→ Parameters 229

→ Dimensions 235

8.1.3.1 Setup

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



On panel **General** you can select:

- **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.

- **Splitter blades**
  Design impeller with or without splitter blades.

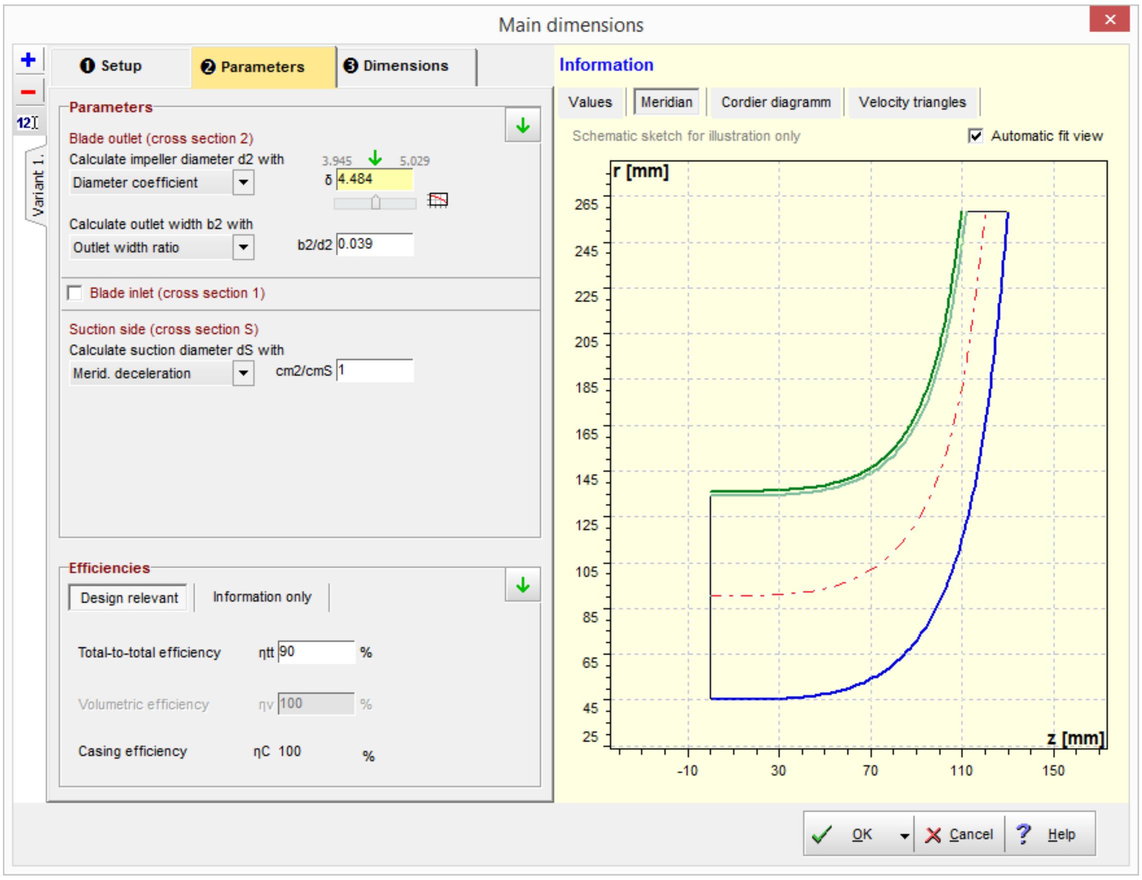
- **Unshrouded**
  Design a shrouded (closed) or unshrouded (open) impeller.
  For an unshrouded impeller you have to define the **tip clearance**.

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 161 for more information.

Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see Global setup 71).

8.1.3.2 Parameters

On page **Parameters** you have to put in or to modify parameters resulting from approximation functions in dependence on specific speed nq or flow rate Q (see Approximation functions 145).

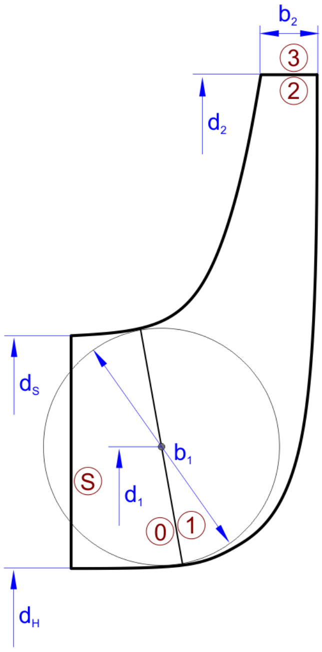


For details of how to handle the parameter edit fields please see Edit fields with empirical functions 47.

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 ψ	▪ dimensionless expression for the specific enthalpy Δh_{is}=Y and Δh=Y_{eff} resp. $\psi = \frac{\Delta h_{is}}{u_2^2/2}$ and $\psi = \frac{\Delta h}{u_2^2/2}$ ▪ high → small d_2, flat characteristic curve low → high d_2, steep characteristic curve ▪ If the check box "**use η**" is set d_2-calculation is done on the basis of $\Delta h = \Delta h_{is}/\eta$. Otherwise Δh_{is} - the isentropic specific enthalpy - is used.
(Total) Flow coefficient φ_t	▪ dimensionless flow rate $\varphi_t = \frac{Q_{t,S}}{\frac{\pi}{4} d_2^2 u_2}$ 0.01 narrow radial impeller, untwisted blades 0.15 mixed-flow impeller, twisted blades
Diameter coefficient δ	▪ according to Cordier diagram (see Dimensions 235)

Machine Mach number Ma_u	▪ dimensionless peripheral speed of impeller related to total inlet speed of sound $Ma_u = \frac{u_2}{a_{t,S}}$
Peripheral speed u_2	▪ Limiting values due to strength as a function of the material

For b_2-calculation

Outlet width ratio b_2/d_2	▪ 0.01...0.15 (with nq rising)
Meridional flow coefficient φ_m	▪ dimensionless flow rate $\varphi_m = \frac{Q_2}{\pi d_2 b_2 u_2} = \frac{c_{2m}}{u_2}$ 0.10...0.50 (with nq rising)

For d_1-calculation (optional)

Diameter ratio d_1/d_2	d_1/d_2=0.3...0.8
Relative deceleration w_2/w_1	w_2/w_1>0.7 or $f(b_2/d_2)$

For b_1-calculation (optional)

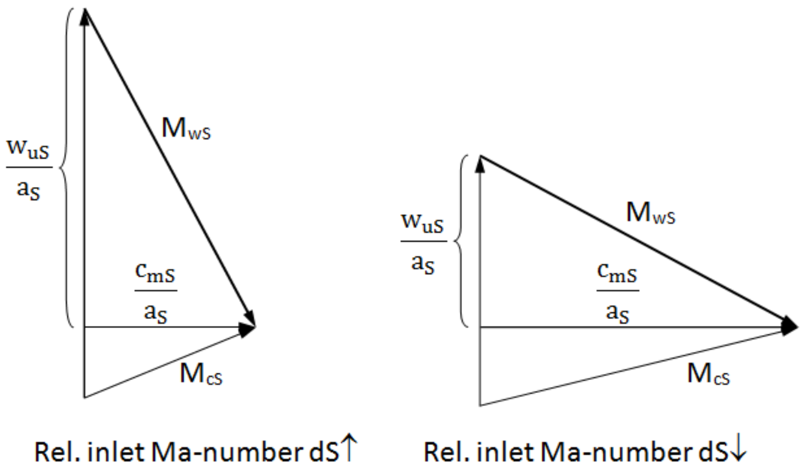
Meridional deceleration c_{m2}/c_{m1}	c_{m2}/c_{m1} = 0.8...1.25

for d_S-calculation

Meridional deceleration c_{m1}/c_{mS} or c_{m2}/c_{mS}	c_{m1}/c_{mS} = 0.9...1.1 c_{m2}/c_{mS} = 0.7...1.3

Relative inlet flow angle β_S	$\beta_S = \arctan\frac{c_{mS}}{w_{uS}} = \arctan\frac{c_{mS}}{u_S - c_{uS}} \approx 30°$
Relative inlet Mach number M_{wS}	$M_{wS} = \frac{w_S}{a_S} = \frac{\sqrt{c_{mS}^2 + w_{uS}^2}}{a_S} \leq 0.75 \ldots 0.85$

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

- flow efficiency η_{tt} (total-total)
- volumetric efficiency η_v

Information only

- mechanical efficiency η_m
- motor efficiency η_{mot}
- casing efficiency η_c (displayed for information only, see Global setup 71)

The casing efficiency η_c is used additionally for impeller dimensioning in order to compensate the flow losses in the casing.

The losses resulting in energy dissipation from the fluid form the **impeller efficiency**.

$$\eta_{Im} = \eta_{tt}\eta_v$$

Impeller, casing 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.

$$\eta_{St} = \frac{P_Q}{P_D} = \eta_{Im}\eta_c\eta_m$$

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	casing	η_C	casing	**yes**: for energy transmission
	impeller	η_{tt}	flow	
		η_V	volumetric	**yes**: for flow rate

	mechanical	η_m	mechanical	**no**: for overall information only
stage incl. motor	electrical	η_{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 145 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:

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

(rising with impeller size)

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

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

(rising with impeller size)

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

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	$P_L = P_D - P_Q = P_D(1-\eta_{St})$
Impeller efficiency	$\eta_{Im} = \eta_{tt}\eta_v$
Stage efficiency	$\eta_{St} = \frac{P_Q}{P_D} = \eta_{Im}\eta_m\eta_c$
Stage efficiency incl. motor	$\eta_{St}^* = \frac{P_Q}{P_{el}} = \eta_{St}\eta_{mot}$
Total-to-static efficiency	$\eta_{ts} = \frac{\pi_t^{\frac{\kappa-1}{\kappa}}\left(1-\frac{c_2^2}{2c_p T_{tS}}\right)-1}{\tau_t - 1}$ (perfect gas model)

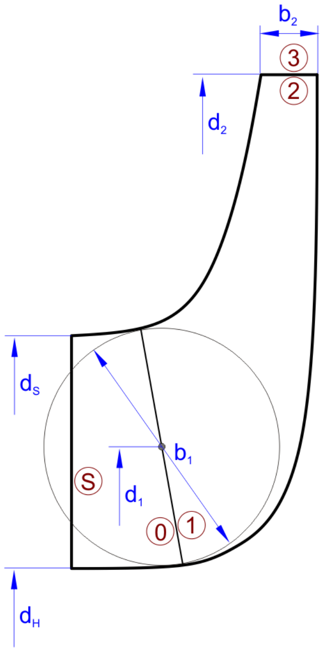
8.1.3.3 Dimensions

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

→ Shaft/Hub 267

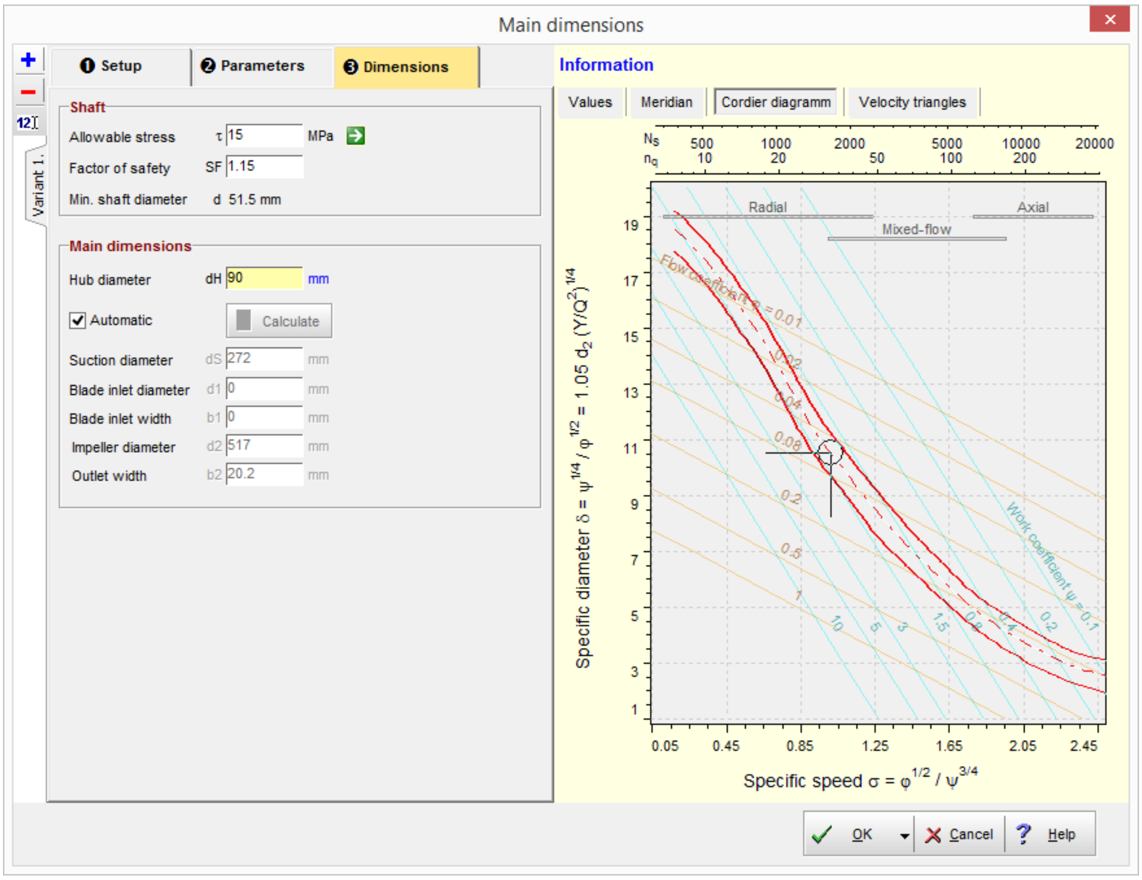
The main dimensions of a designed impeller - suction diameter d_S, impeller diameter d_2, outlet width b_2 - can be seen on **Main dimensions** panel. They can be recomputed by pressing the **Calculate**-button. The computation is based on "Euler's Equation of Turbomachinery", on the continuity equation and the relations for the velocity triangles as well as on the parameters and parameter ratios given in the tab sheets **Setup** and **Parameters**.

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



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

Regarding the impeller size one should try to attain d_2 values as low as possible. But there is a limit for a specified task: lower impeller diameters are leading to higher blade loading - up to blade angles β_2 which may not be suitable anymore.



Information

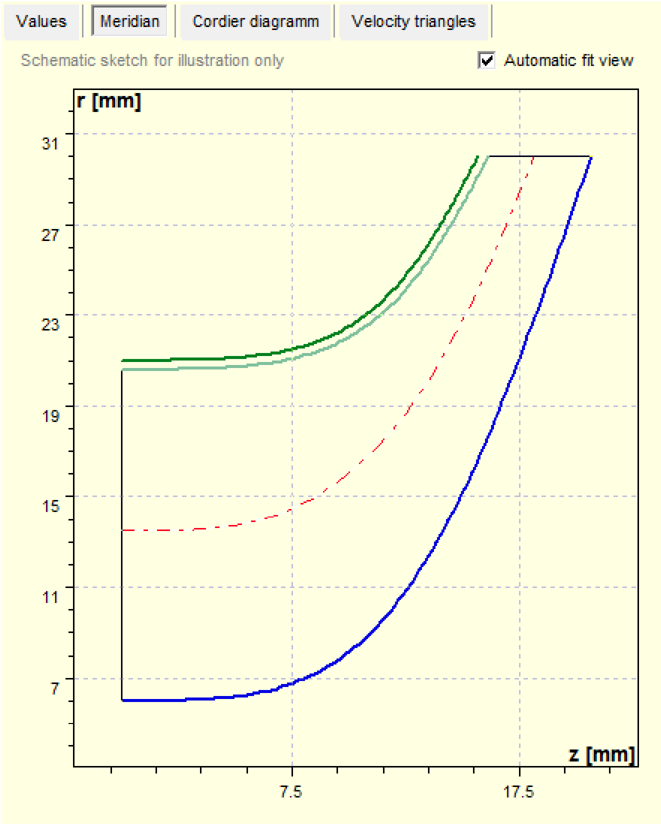
In the right panel of any tab sheet an **information** panel is situated, which holds the computed variables in accordance to the actual state of design, the resulting Meridional section 238 as well as the Cordier-Diagramm 239 with the location of the best point. These three sections can be chosen by 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:

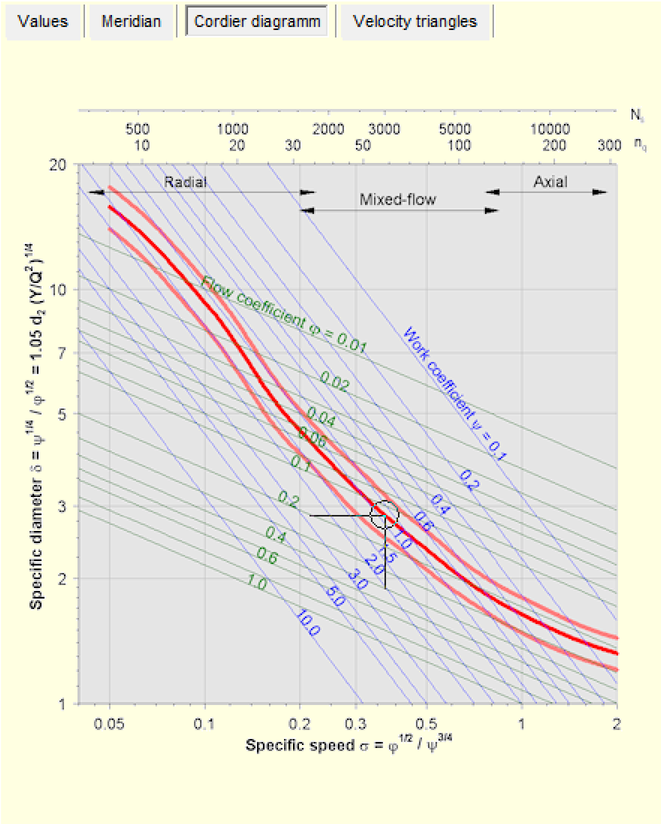
Work coefficient	
Flow coefficient	

Meridional flow coefficient	$\varphi_m = \dfrac{Q_2}{\pi d_2 b_2 u_2} = \dfrac{c_{m2}}{u_2} = 0.1 \ldots 0.5$
Diameter coefficient	$\delta = \dfrac{\psi^{1/4}}{\varphi_t^{1/2}} = 1.05 d_2 \left(\dfrac{Y}{Q_{tS}^2}\right)^{1/4}$
Tangential force coefficient	$c_t = \dfrac{\psi}{\eta_{tt}\,\varphi_m} \approx 3 \ldots 6$
Outlet width ratio	b_2/d_2 = 0.01...0.15
Diameter ratio	d_S/d_2
Inlet Mach number	$Ma_{wS} = \dfrac{\sqrt{w_{mS}^2 + w_{uS}^2}}{\sqrt{\kappa R Z T_S}} \le 0.75 \ldots 0.85$
Outlet Mach number	$Ma_{c2} = \dfrac{1}{\sqrt{\left(\dfrac{a_{t,2}}{c_2}\right)^2 - \dfrac{\kappa - 1}{2}}} \le 1$ (perfect gas model)
Reaction	$r = 1 - \dfrac{c_2^2}{2Y}$
thermodynamic values for - impeller inlet (cross section S) - impeller outlet (cross section 2)	ρ, p, T, c_m, c_u, w, u

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



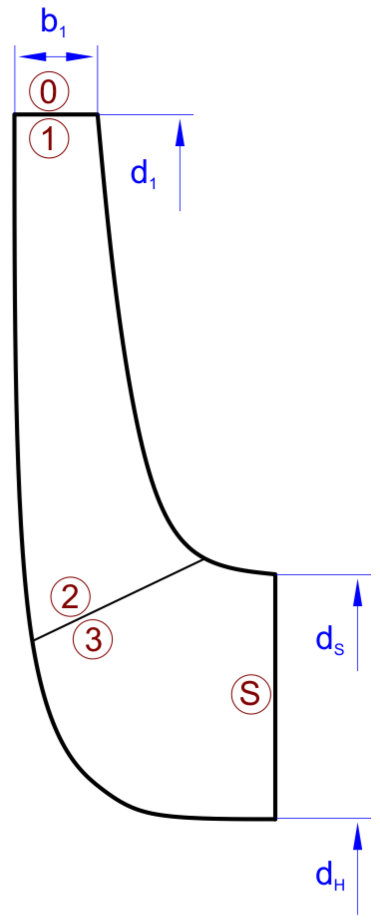
The **Cordier diagram** is based on an intensive empirical analysis of proved turbomachinery using extensive experimental data.



8.1.4 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 242

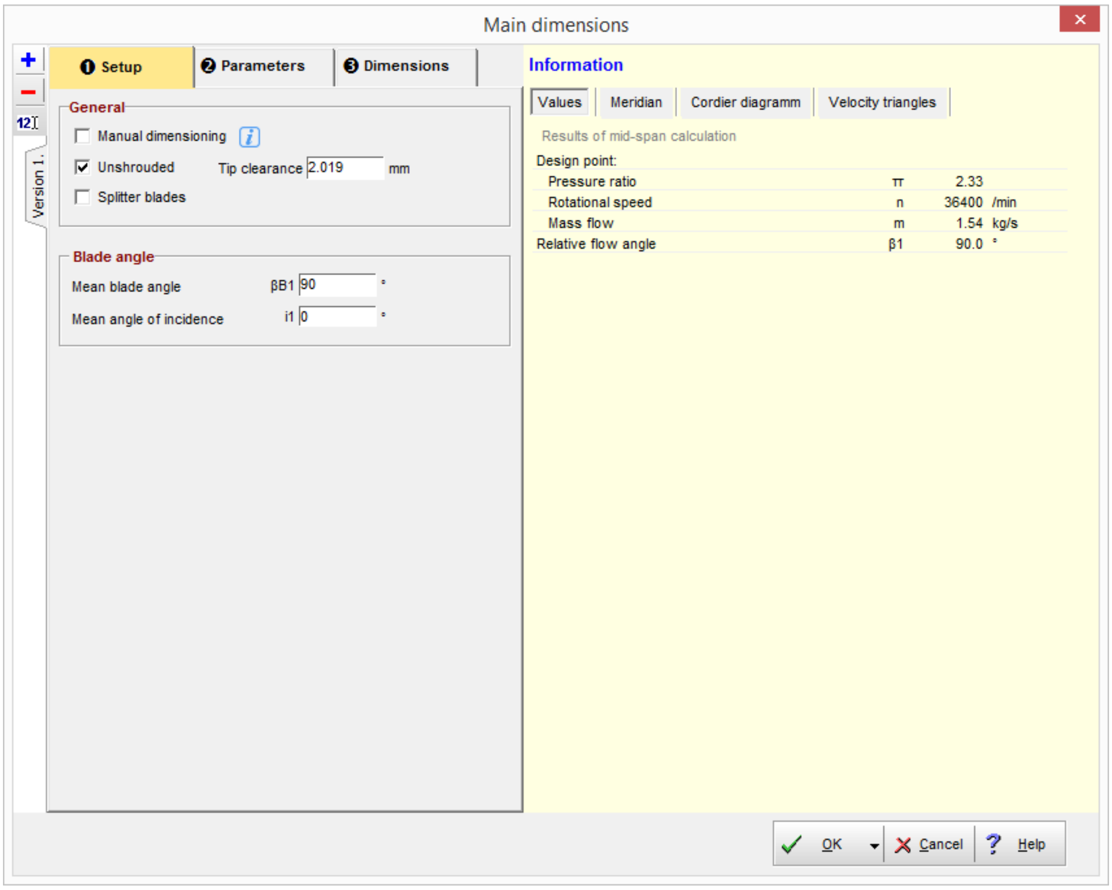
→ Assumptions 243

→ Dimensions 243

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.

8.1.4.1 Setup

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



On panel **General** you can select:

- **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.

- **Splitter blades**
  Design the rotor with or without splitter blades.

- **Unshrouded**
  Design a shrouded (closed) or unshrouded (open) rotor.
  For an unshrouded rotor you have to define the **tip clearance**.

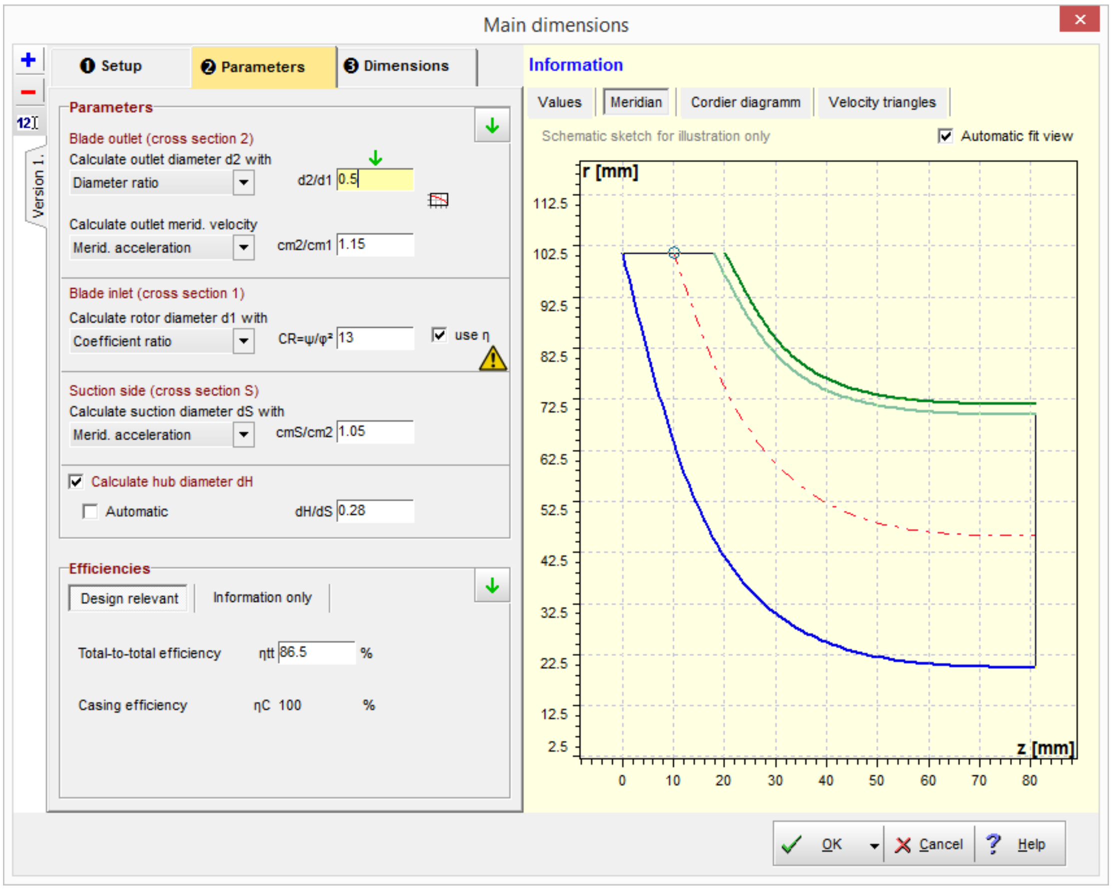
When creating a new design the initial default settings for some important properties are displayed in the panel **Initial default settings**. These settings are used in further design steps and can be modified by selecting the **Change settings** button. Of course these default settings can be modified manually in the appropriate design steps. See Preferences: Impeller/ Stator settings 161 for more information.

The design concept is based on a mean flow area, therefore a mean blade angle bB1 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°.

Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see Global setup71).

8.1.4.2 Parameters

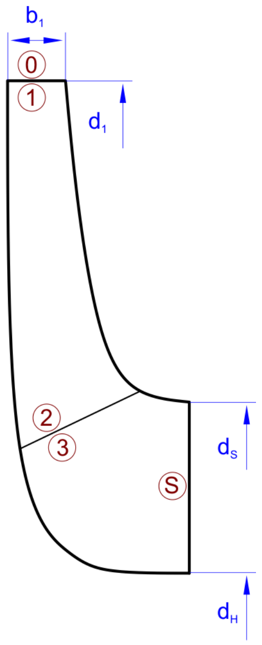
On page **Parameters** one has to put in or to modify parameters resulting from approximation functions in dependence on specific speed nq (see Approximation functions145).



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 47.

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

Work coefficient ψ	▪ dimensionless expression of the specific enthalpy $\psi = \frac{\Delta h_{is}}{u_2^2/2}$ and $\psi = \frac{\Delta h}{u_2^2/2}$ ▪ big → small d_1 small → big d_1 ▪ Guideline ~ 2 ▪ If the check box "**Use η**" is set d_1-calculation is done on the basis of $\Delta h = \Delta h_{is} \cdot \eta$. Otherwise Δh_{is} - the isentropic specific enthalpy - is used.
Flow coefficient φ_m	▪ dimensionless mass flow ▪ in accordance to Cordier-Diagramm 251

Tangential force coefficient $c_t = \psi/\varphi_m$	▪ Coefficient of a flow force pointing in tangential direction 3 … 4 Francis high-speed turbine 4 … 8 Normal-speed turbine 8 …10 Low-speed turbine
Coefficient ratio $c_R = \psi/\varphi_m^{\ 2}$	▪ Ratio of work to the square of the meridional speed 6 …10 Francis high-speed turbine 10…12 Normal-speed turbine 12…30 Low-speed turbine

Between the work coefficient ψ the relative flow angle β_1 and the tangential force coefficient ψ/φ_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 β_1= 90° the work coefficient becomes ψ=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_N/d_S	~0.3

There are three specification modes of the diameter ratio d_H/d_S:

- Direct input

- Automatic calculation: option "Automatic". Here the diameter ratio will be adjusted in a way that the guideline of the geometrical ratios [251] will be met.

- Direct specification of d_H in the tab sheet **Dimensions**. Here the diameter ratio is not necessary.

With diameter ratio d_S/d_1 option "Automatic" is deactivated.

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 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:

(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	calculated with total density in the outlet: $Q_t = \frac{\dot{m}}{\rho_{t2}}$
Total pressure inlet p_{t1}	$p_{t1} = \pi \cdot p_{t2}$
Pressure ratio total-total	π_{tt}
Pressure ratio total-static	π_{ts}
Stage efficiency total-total	η_{ttSt}
Efficiency total-static	η_{ts}
Isentropic velocity ratio	$\nu_{ts} = \frac{u_1}{\sqrt{2\Delta h_{is}}}$

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.

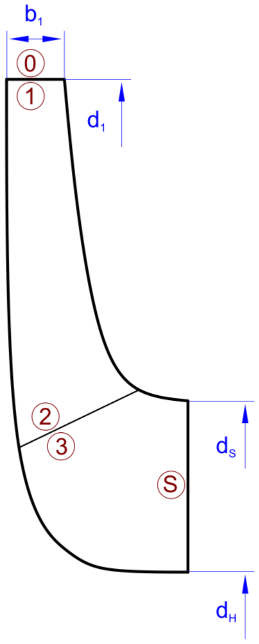
8.1.4.3 Dimensions

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

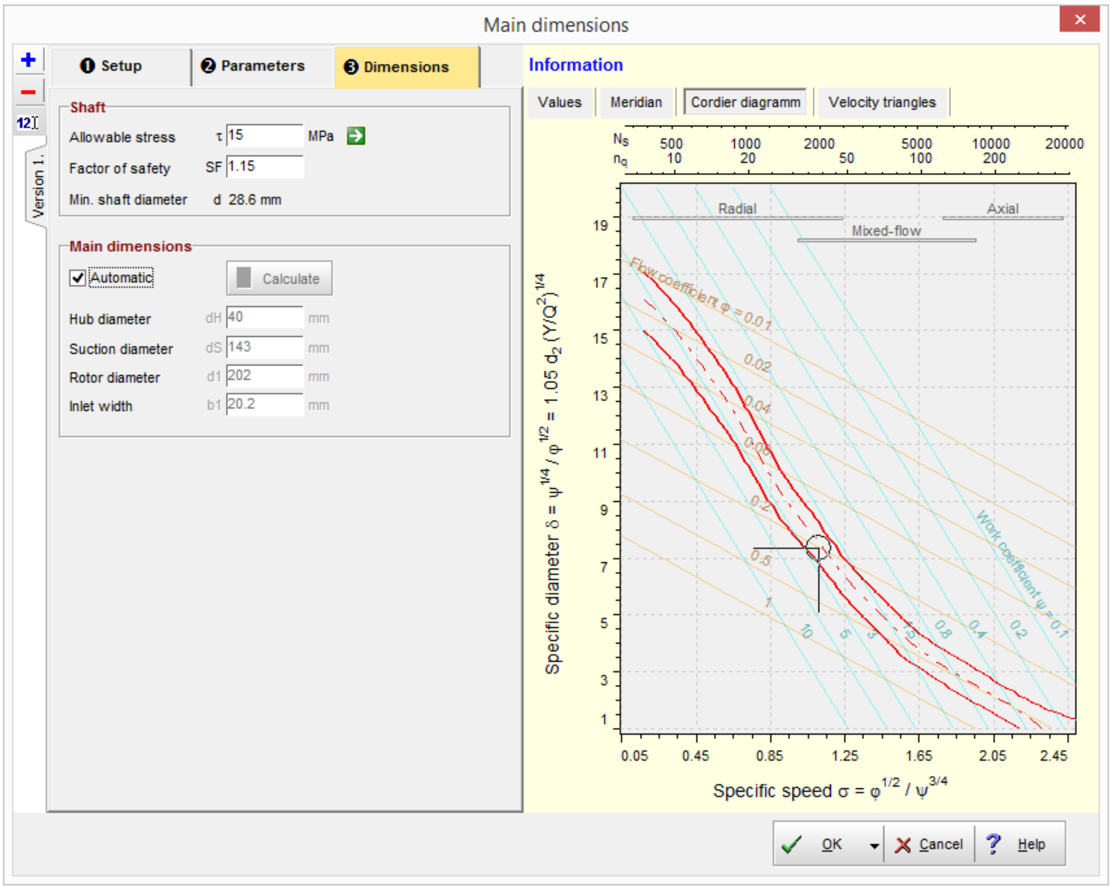
➜ Shaft/ Hub 267

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**.

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



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



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 251 as well as the Cordier-Diagramm 251 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**:

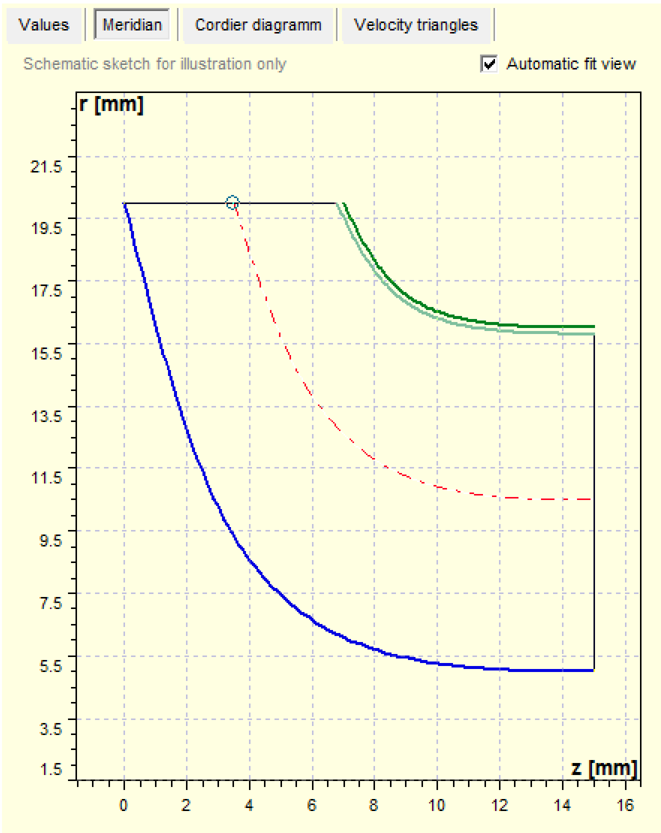
Work coefficient		
Flow coefficient		

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 159)	$n_q = n\left[min^{-1}\right]\frac{\sqrt{Q\left[m^3/s\right]}}{\left(Y\left[\frac{m^2}{s^2}\right]\frac{1}{g}\right)}$	points to machine type and general shape of rotor
Inlet pressure, density and temperature	p_1, T_1, ρ_1, p_{t1}, T_{t1}, ρ_{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	b_1^*	
Outlet pressure, density and temperature	p_2, T_2, ρ_2, p_{t2}, T_{t2}, ρ_{t2}	static and total values
Outlet velocities	c_2, c_{u2}, c_{m2}, w_2	
Peripheral speed at outlet	$u_2 = \pi \cdot d_2 \cdot n$	
Outlet Ma-Number	$M_2 = \frac{c_2}{a_2}$	
Mean diameter at outlet		
Width at outlet		

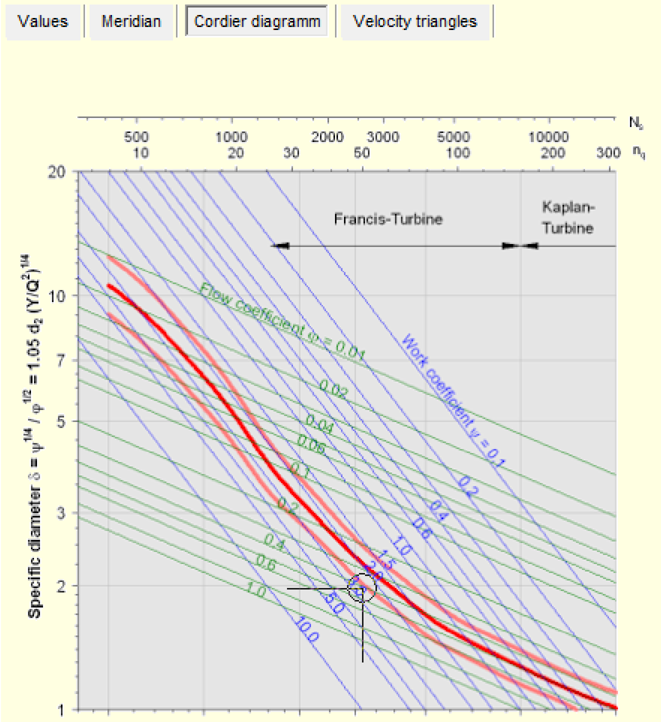
Ratio Width-diameter at inlet	b_1/d_1	guideline: 0.05..0.15
Diameter ratio	d_2/d_{2min} with: $d_{2min} = \sqrt{\frac{1}{2}\left(d_S^2 - d_N^2\right)}$	guideline: 1.005..1.05
Ratio radius-width at outlet	$\frac{(r_S - r_N)}{b_2} = \frac{(d_S - d_N)}{2 \cdot b_2}$	guideline: 1.005..1.05

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

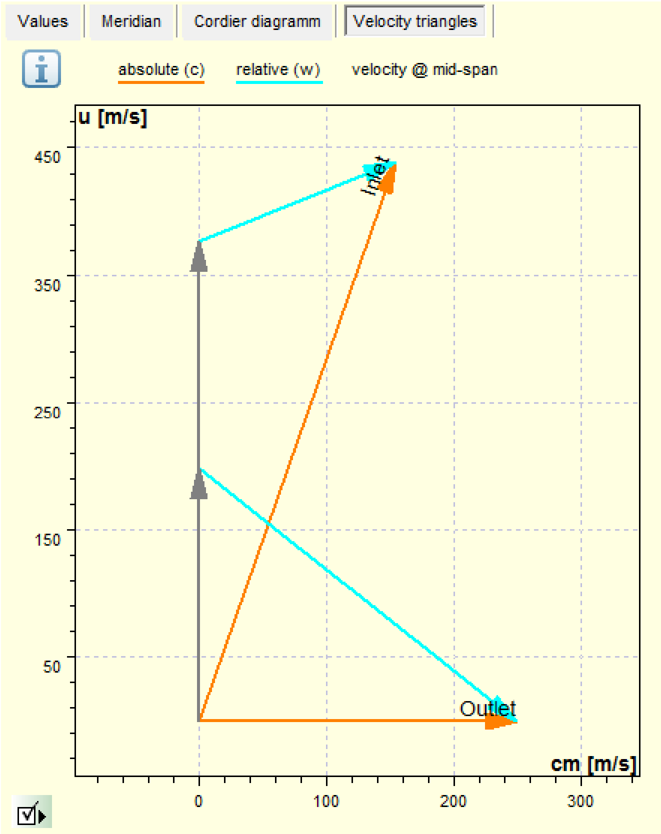
The Meridional preview is based on the main dimensions designed until this point.



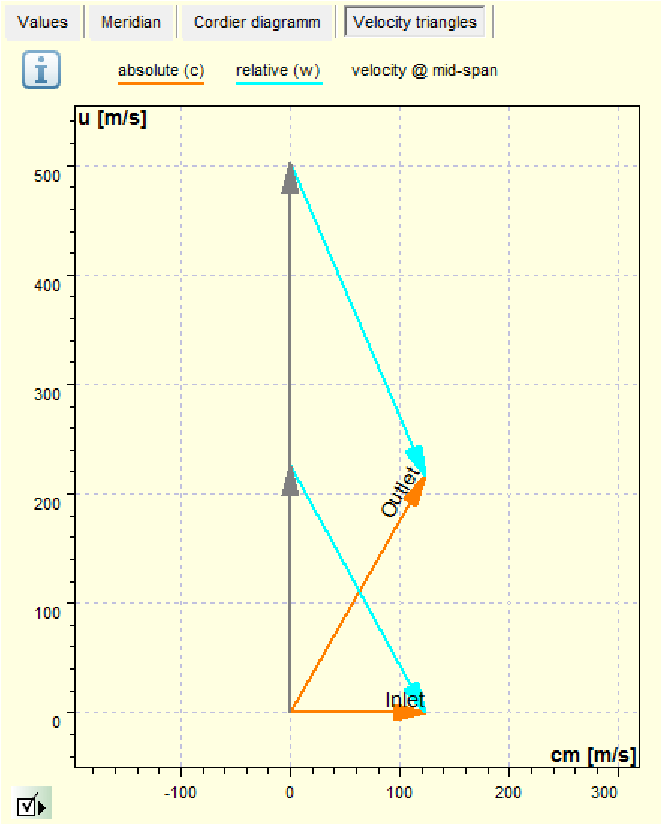
The **Cordier diagram** is based on an intensive empirical analysis of proved turbomachinery using extensive experimental data.



The **Velocity triangles** are the result of a mid-span calculation and are based on the [design point](71) 71 and the main dimensions.



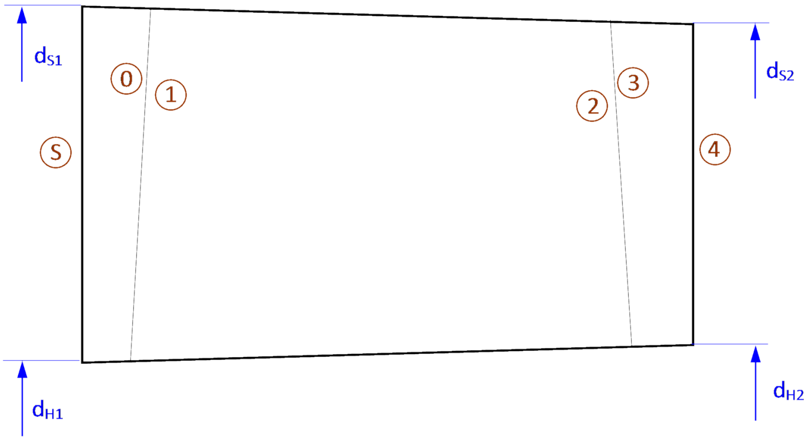
The **Velocity triangles** are the result of a mid-span calculation and are based on the design point 71 and the main dimensions.



8.1.5 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

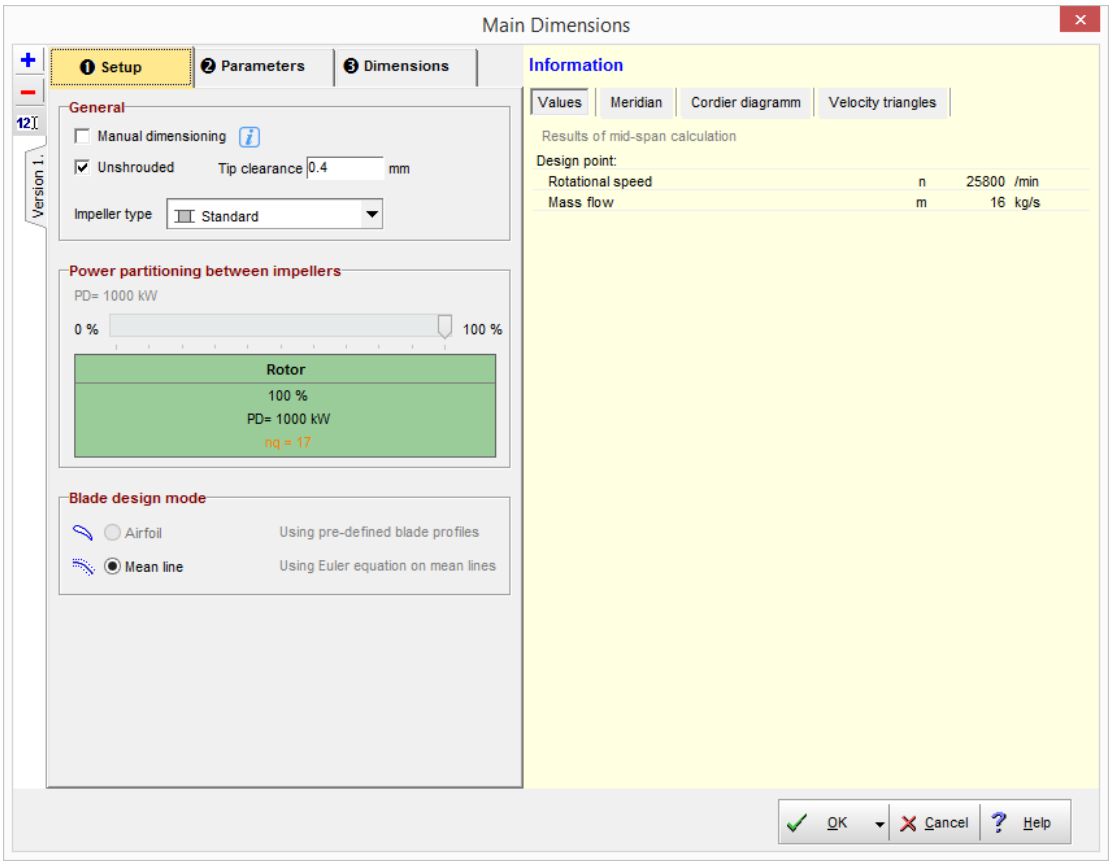
→ Setup 256

→ Parameters 258

→ Dimensions 261

8.1.5.1 Setup

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



On panel **General** you can select:

- **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**.

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

In case more than 1 rotor is contained in the project the design point 71 (Power output, pressure ratio) can be distributed amongst the rotors using the power partitioning. The energy goal used for the design of the selected rotor (index i) is determined by:

,

where the P is the actual power output. The lower case e_i is the ratio describing the power partitioning for the selected rotor.

On panel **Blade design mode** currently one design mode is available:

- **Mean line** 292
  Design using Euler's equation on mean lines.

In case a pressure ratio π has been specified in the Global setup 71 the pressure ratio used for the design of the selected rotor is determined by:

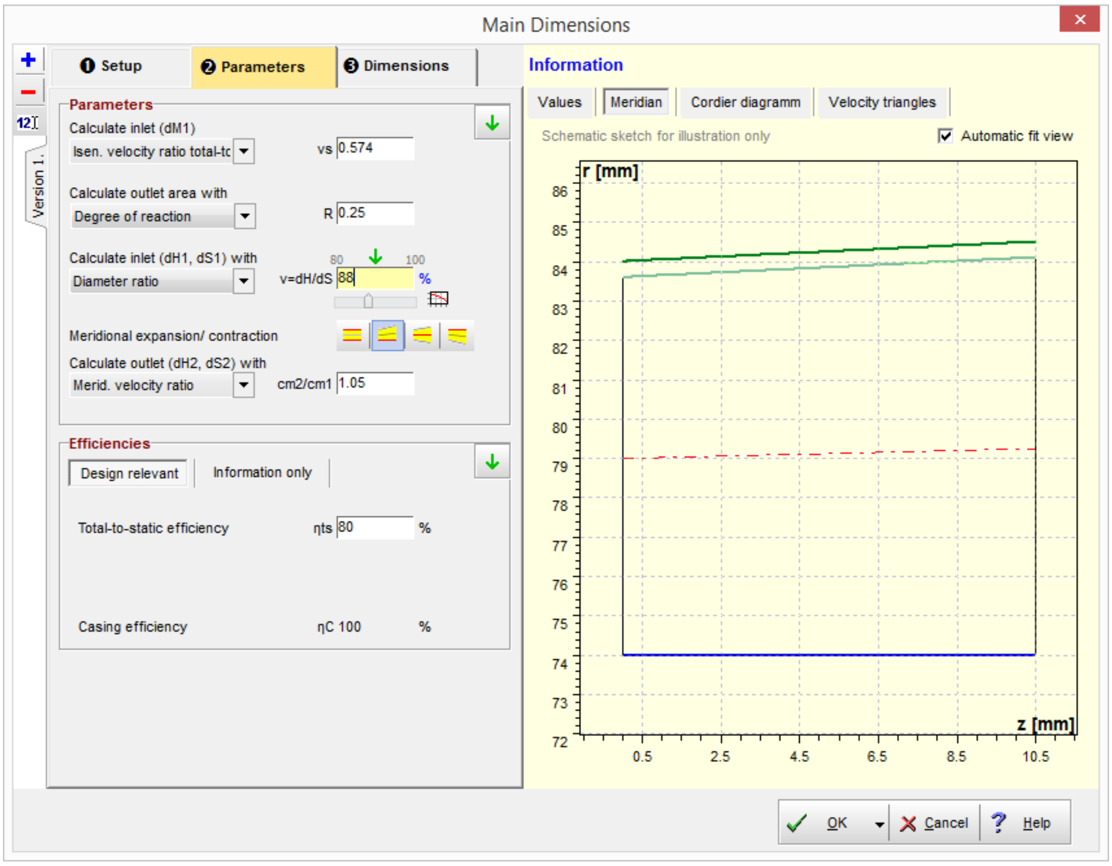
$$\pi_i = \frac{\pi}{\prod_{j \neq i} \pi_j}.$$

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 161 for more information.

Some design point values are displayed in the right **Information** panel when selecting the page **Values** (see Global setup 71).

8.1.5.2 Parameters

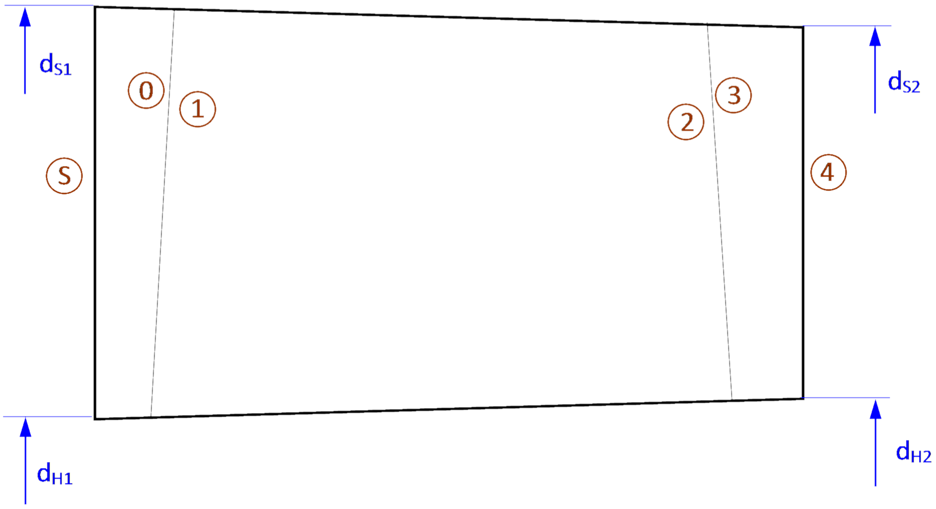
On page **Parameters** one has to put in or to modify parameters resulting from approximation functions in dependence on specific speed nq (see Approximation functions 145).



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 details of how to handle the parameter edit fields please see Edit fields with empirical functions 47.

With the help of the following parameters the inlet of the rotor can be calculated.

Isentropic velocity ratio ν_{is}	Mean inlet diameter $0.5(d_{S1}+d_{H1})$ $$\nu_{ts} = \frac{u_1}{\sqrt{2\Delta h_{ttis}}} = \frac{\pi \cdot n \cdot d_{M1}}{\sqrt{2\Delta h_{ttis}}}$$
Degree of reaction R	Outlet tip diameter d_{S2} (and via c_{m2}/c_{m1} d_{H2}) $$R = \frac{\Delta h}{\Delta h_{tt}}$$
Tangential abs. velocity component c_{u2}	Outlet tip diameter d_{S2} (and via c_{m2}/c_{m1} d_{H2})
Diameter ratio d_H/d_S	Inlet hub diameter d_{H1}

The outlet section can be calculated with:

Meridional velocity ratio c_{m2}/c_{m1}	0.9..1.1 strictly axial $d_{H2} = d_{H1}$ and $d_{S2} = d_{S1}$ const. hub $d_{H2} = d_{H1}$ const. mid $d_{M2} = d_{M1}$ const. shroud $d_{S2} = d_{S1}$

Efficiency

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

Design relevant

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

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 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:

(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	π_{ts}
Efficiency total-total	η_{tt}
Efficiency total-static	η_{ts}
Isentropic velocity ratio	$\nu_{ts} = \frac{u_1}{\sqrt{2\Delta h_{ttis}}}$

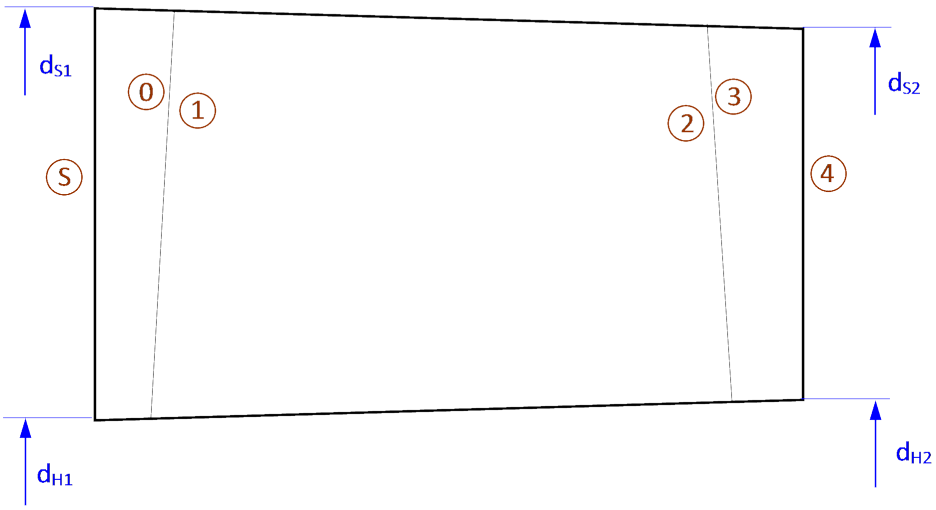
In general for cost reasons single-stage & single-intake machines are preferred covering a range of about 10 < nq < 400. If especially high specific speed values (nq > 400) do occur one can reduce rotational speed n or mass flow rate ? if feasible. Another option would be to operate several single-stage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

Please note: CFturbo® is preferably used between **100 < nq < 400** – axial rotors.

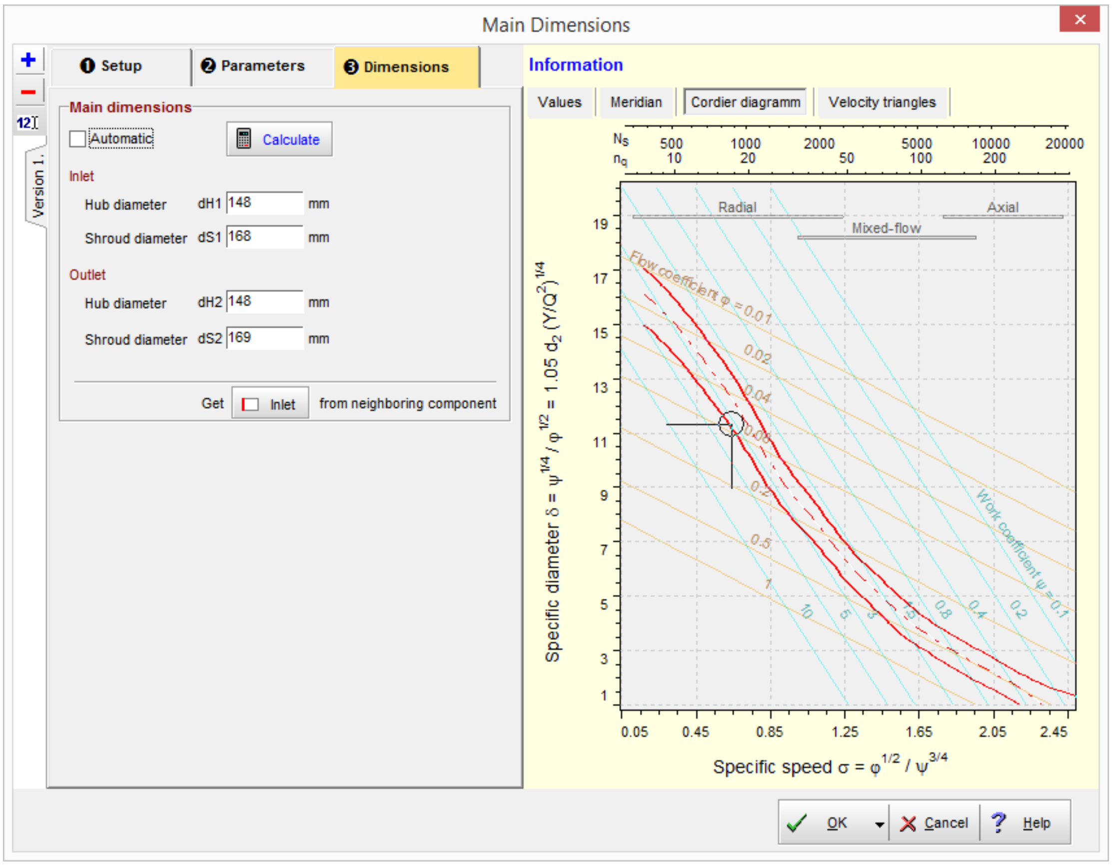
8.1.5.3 Dimensions

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**.

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



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



Information

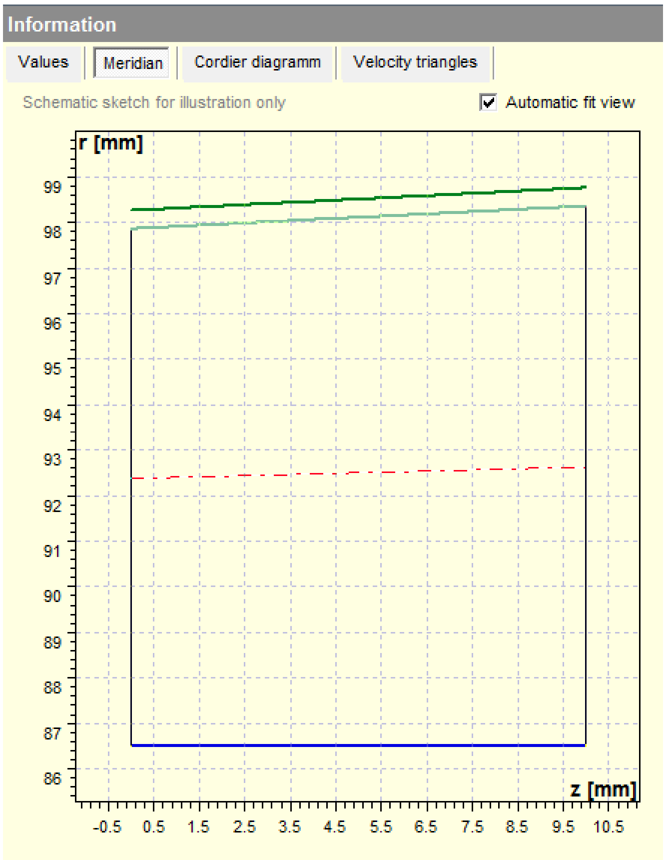
In the right panel of any tab sheet an information panel is situated, which holds the computed variables in accordance to the actual state of design, the resulting Meridional section 251 as well as the Cordier-Diagramm 251 with the location of the best point. These three sections can be chosen by the appropriate soft buttons in the heading.

In the information section of the tab sheet **Dimensions** the following variables are displayed for **Information**:

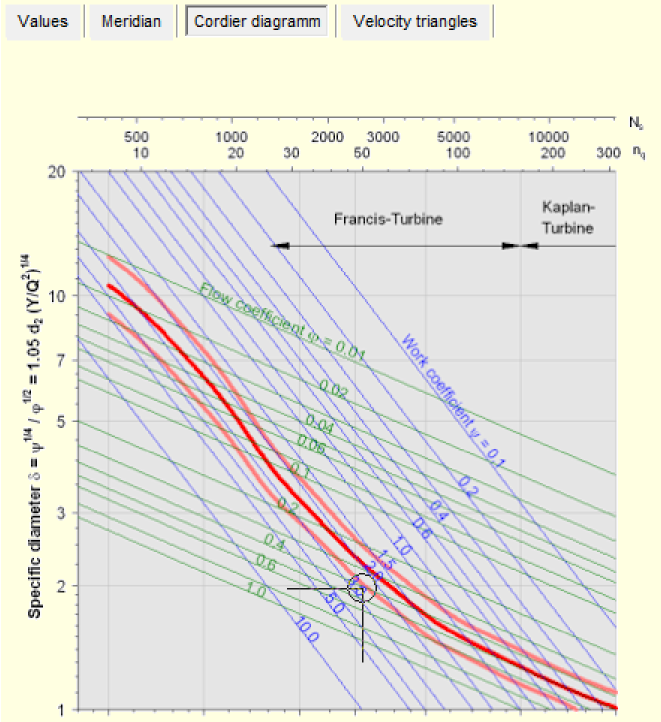
Work coefficient		
Flow coefficient		

Meridional flow coefficient	$\varphi_m = \frac{Q_1}{\frac{\pi}{4}\left(d_{1S}^2 - d_{1H}^2\right)u_1} =$	
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, ρ_1, p_{t1}, T_{t1}, ρ_{t1}	static and total values
Inlet velocities	c_1, c_{u1}, c_{m1}, w_1, u_1	
Inlet Mach-number	$M_1 = \frac{u_1}{a_1}$	
Outlet pressure, density and temperature	p_2, T_2, ρ_2, p_{t2}, T_{t2}, ρ_{t2}	static and total values
Outlet velocities	c_2, c_{u2}, c_{m2}, w_2, u_2	
Outlet Ma-Number	$M_2 = \frac{c_2}{a_2}$	

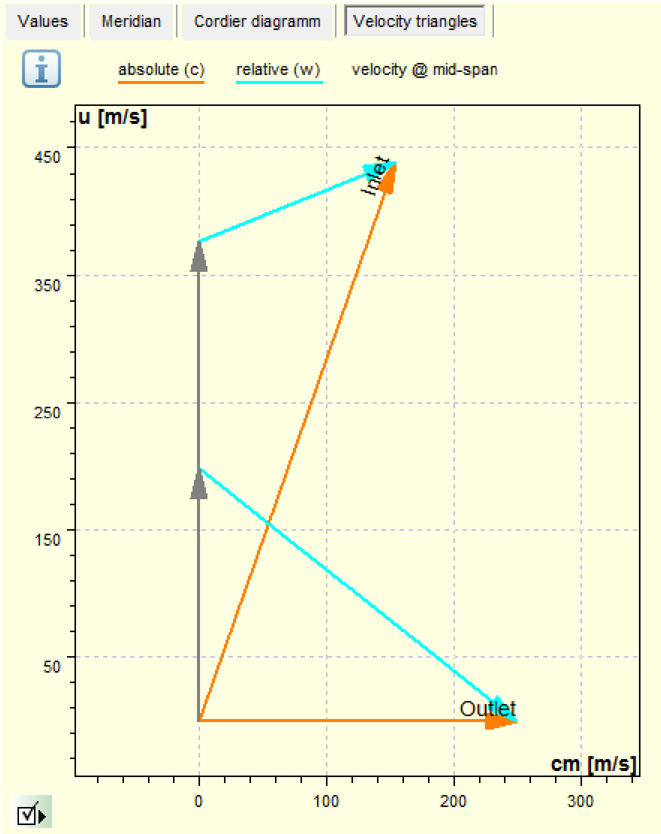
The Meridional preview is based on the main dimensions designed until this point.



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



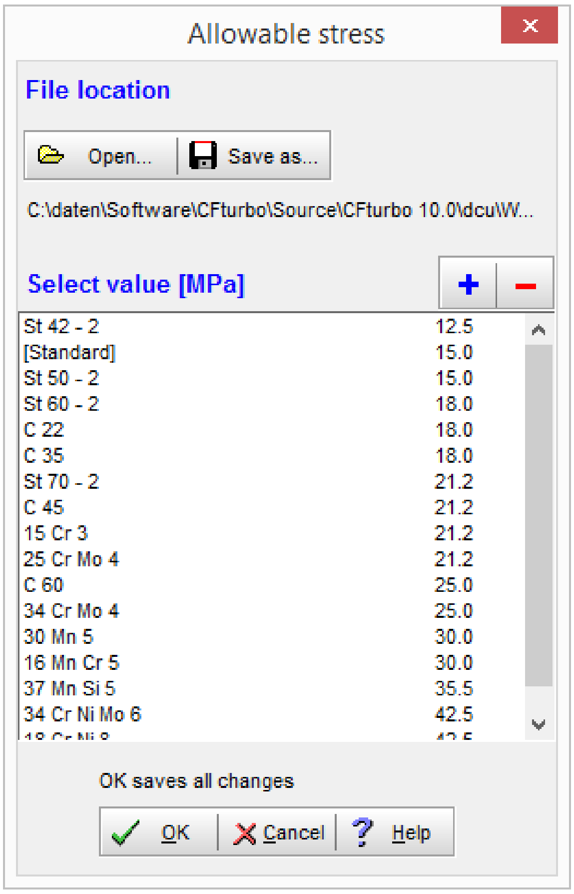
The **Velocity triangles** are the result of a mid-span calculation and are based on the [design point](71) 71 and the main dimensions.



8.1.6 Shaft/Hub

Dimensioning of the shaft diameter is made under application of strength requirements. It is a result of torque M=P/ω 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.cftst** and is located in the installation directory of CFturbo. Modifications of the list will be saved if the user is leaving the dialog window by clicking the **OK**-button. In case there are no write permissions the user can choose another directory to save the file. Renaming of files is possible by **Save as**-functionality. By clicking the **Open**-button a previously saved file can be opened.

To consider a higher load, e.g. due to operating conditions away from the design point, a safety factor SF may be specified leading to a modified proposed shaft diameter d.

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

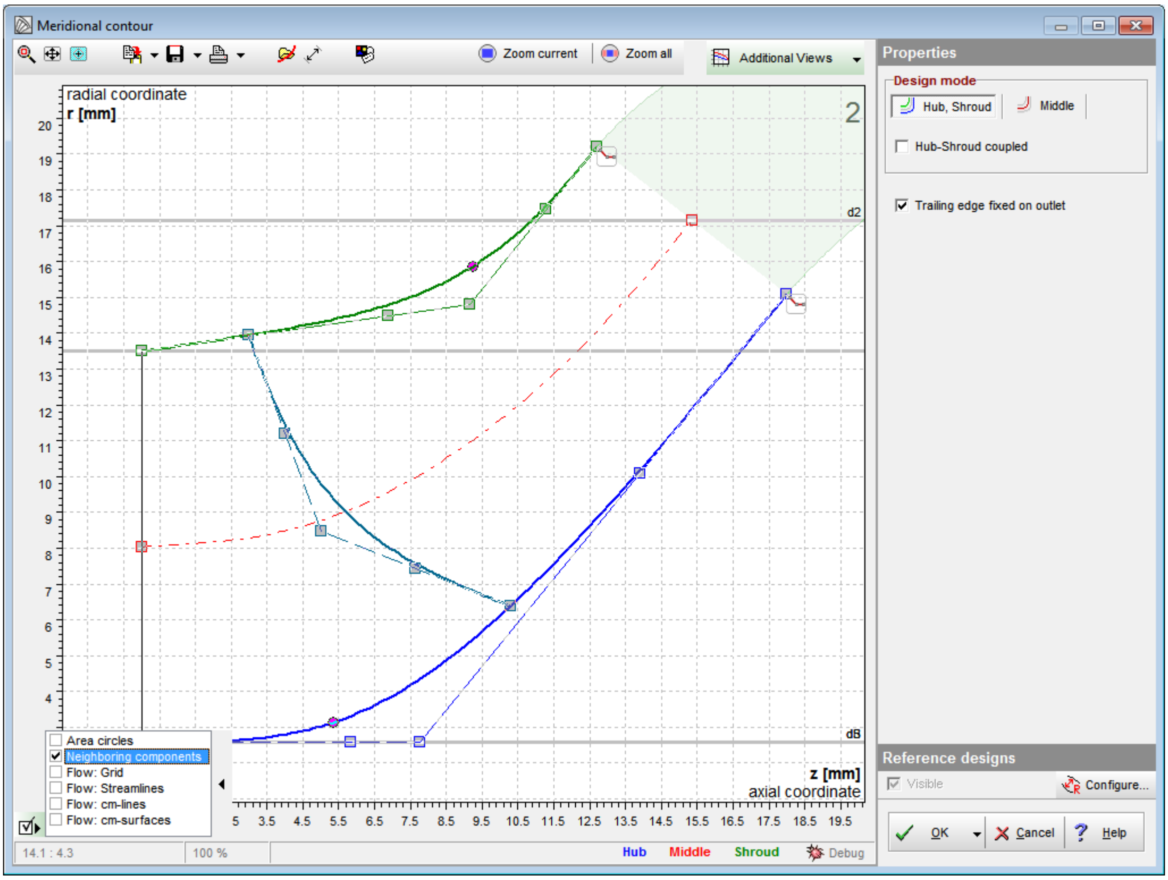
The hub diameter d_H is usually selected as small as possible and depends on the kind of connection of hub and shaft.

8.2 Meridional contour

? Impeller | Meridional contour

The design of the meridional contour is the second important step to design the impeller.

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.



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 or in coupled mode. If the **Hub-Shroud Coupled** check box is checked hub and shroud will be modified simultaneously considering the same relative positions of the Bezier points.

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 46.

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:

$$A = A_{in} + A_{rel}\left(A_{out} - A_{in}\right)$$

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.

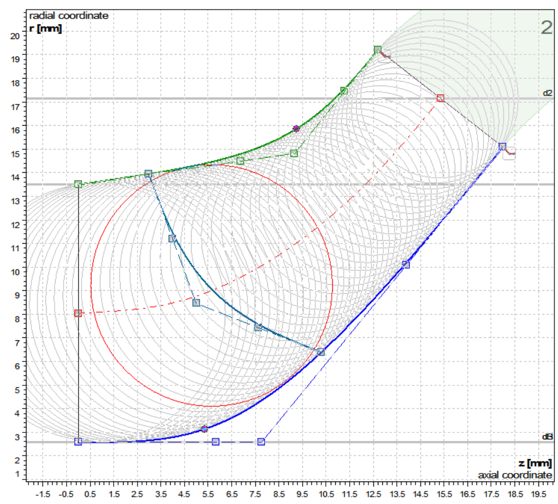
Trailing edge fixed on ...

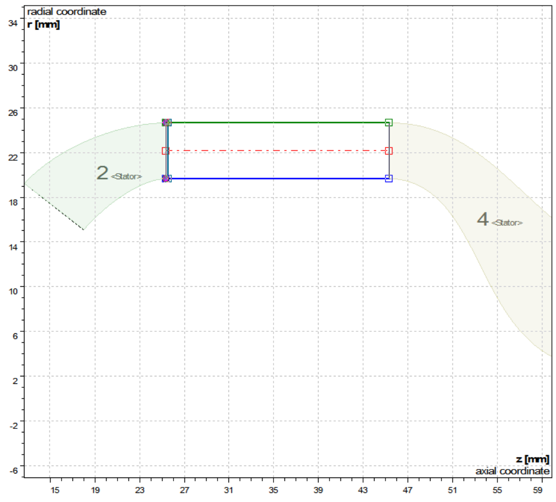
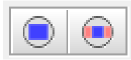
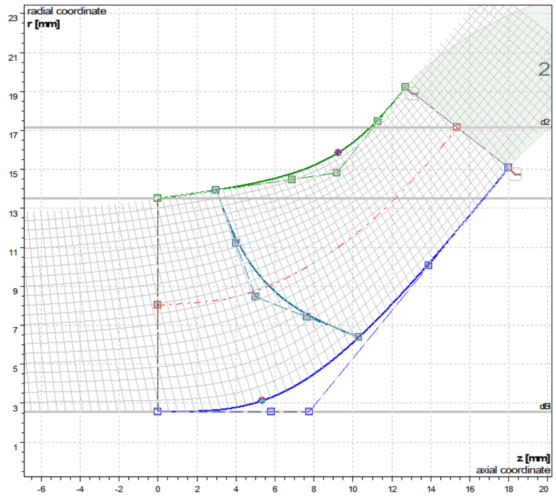
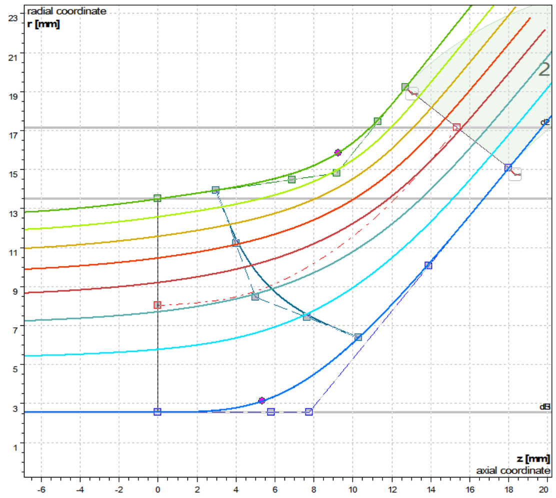
The trailing edge (turbines: leading edge) is fixed on meridional outlet (turbines: inlet) and can not designed like the leading edge 284 (turbines: trailing edge 284).

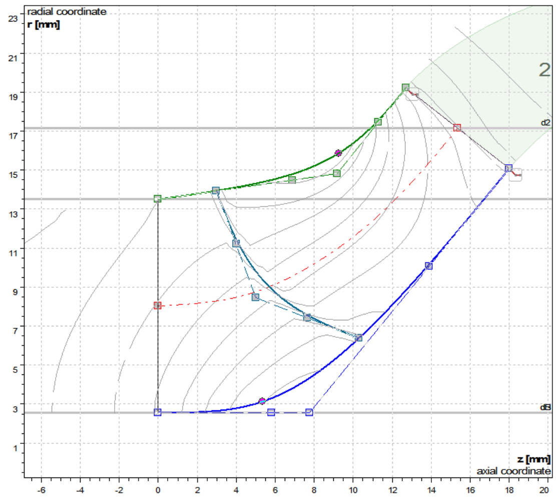
Uncheck this option to detach the trailing edge (turbines: leading edge) from meridional outlet (turbines: inlet) and design its position and shape independently.

Display Options

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

	**Area circles** for calculation of cross section area

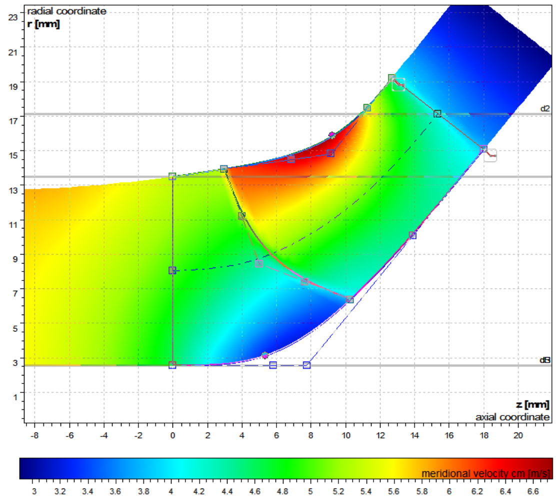
	**Neighboring components** on inlet and outlet side are displayed for information.  Use the buttons to zoom the current meridional shape only or the entire geometry.
	**Meridional flow/ Grid** grid used for meridional flow calculation ➜ see Meridional flow calculation 288
	**Meridional flow/ Streamlines** meridional streamlines, equal mass flow fraction between neighboring streamlines ➜ see Meridional flow calculation 288



Meridional flow/ cm lines

iso lines of const. meridional velocity c_m

➜ see Meridional flow calculation 288



Meridional flow/ cm surfaces

iso surfaces of const. meridional velocity c_m

(scaling is displayed below the diagram)

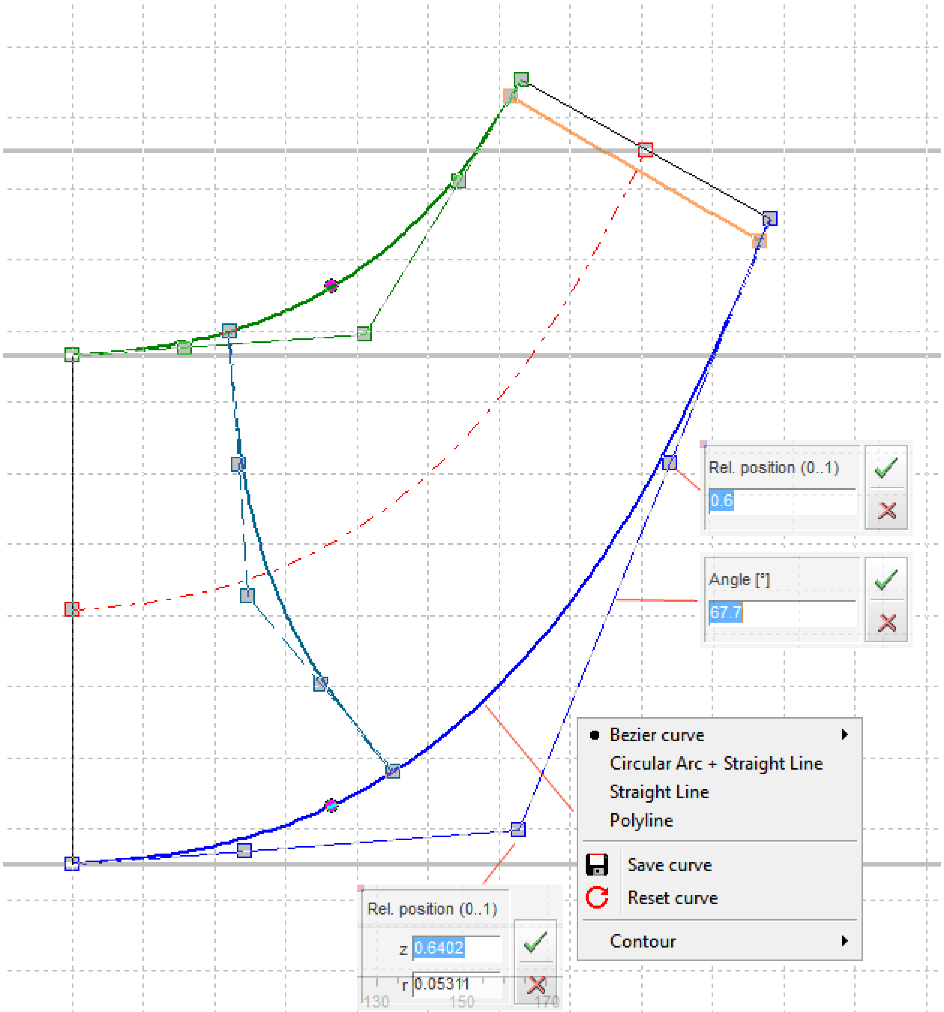
➜ see Meridional flow calculation 288

Possible warnings

Problem	Possible solution
Inlet hub diameter: the deviation between meridional geometry and main dimension is higher 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 190 or the imported curve.

Problem	Possible solution
Inlet shroud diameter: the deviation between meridional geometry and main dimension is higher 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 190 or the imported curve.
Outlet diameter: the deviation between meridional geometry and main dimension is higher 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 190 or the imported curve.
Outlet width: the deviation between meridional geometry and main dimension is higher 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 190 or the imported curve.

8.2.1 Hub-Shroud contour



Hub & Shroud

Hub and shroud countours can be designed as:

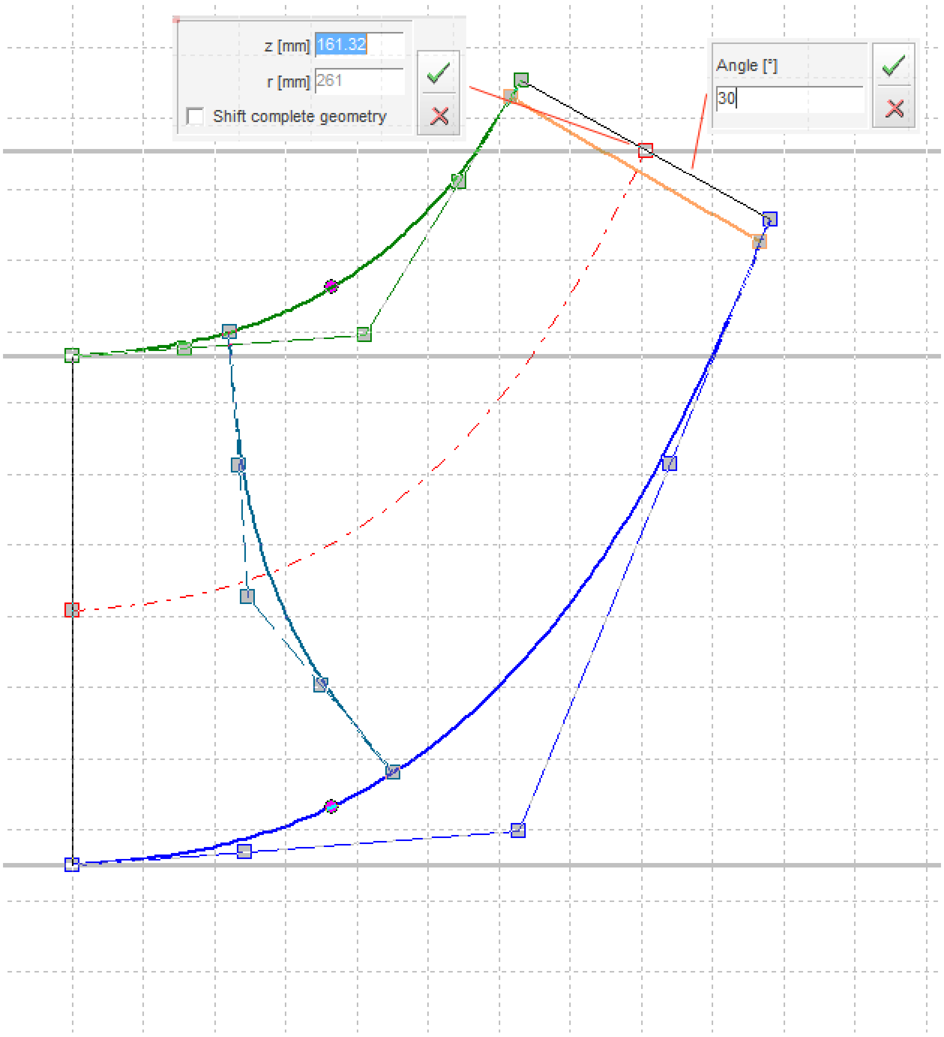
- **Bezier curve**
  The curve is defined by the position of the Bezier points.
  ➜ Details 276

- **Circular Arc + Straight line**
  The curve consists of a circular arc and a straight line.
  ➜ Details 280

- **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**).



Special context menu features

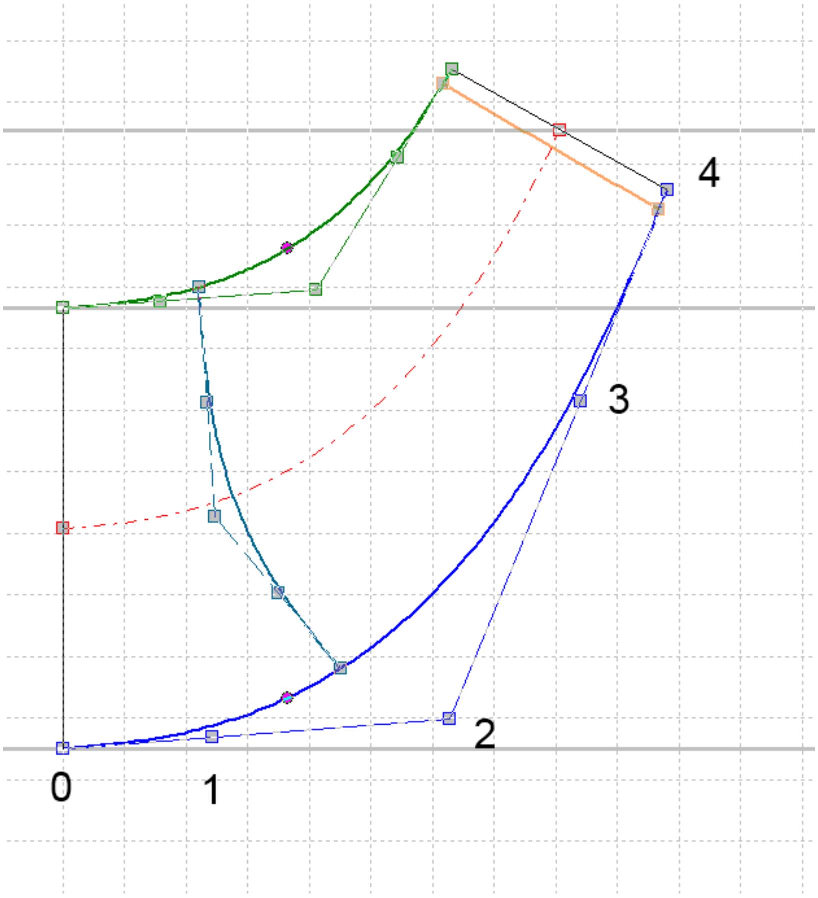
- On the endpoints of hub and shroud the complete geometry can be shifted optionally (Shift complete geometry). Hence the geometry can be positioned on a specific axial position.
- There are some reasonable constraints when working in this simplified mode e.g. the inclination angle of the trailing edge can only be set when hub and shroud are in Bezier mode both.

8.2.1.1 Bezier

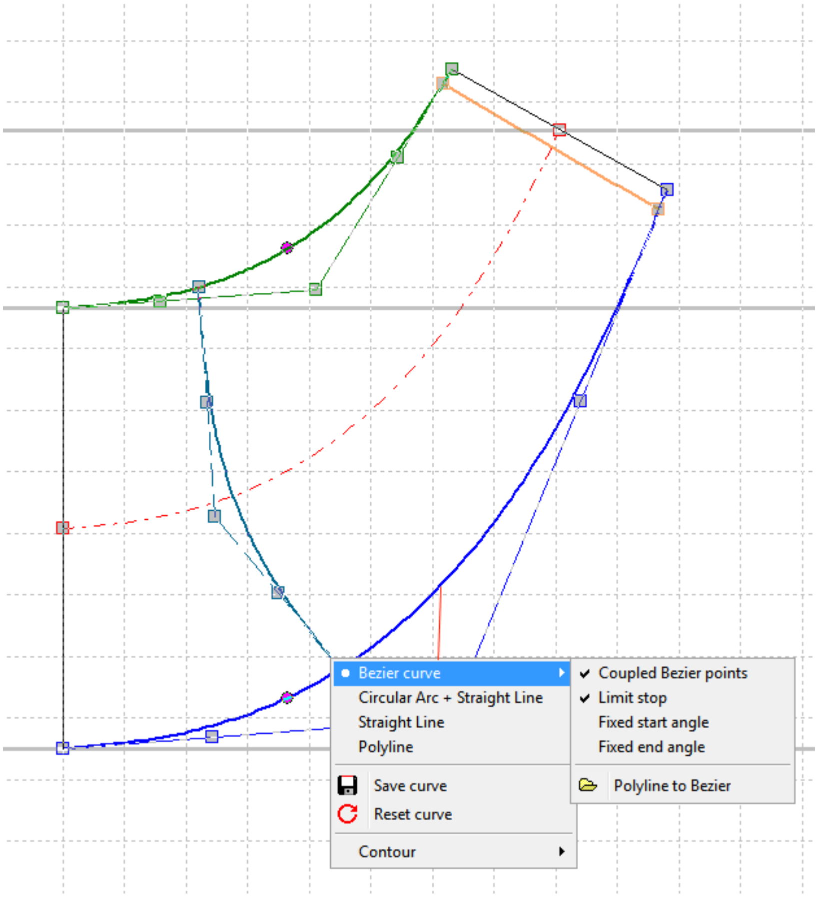
Bezier curves

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.

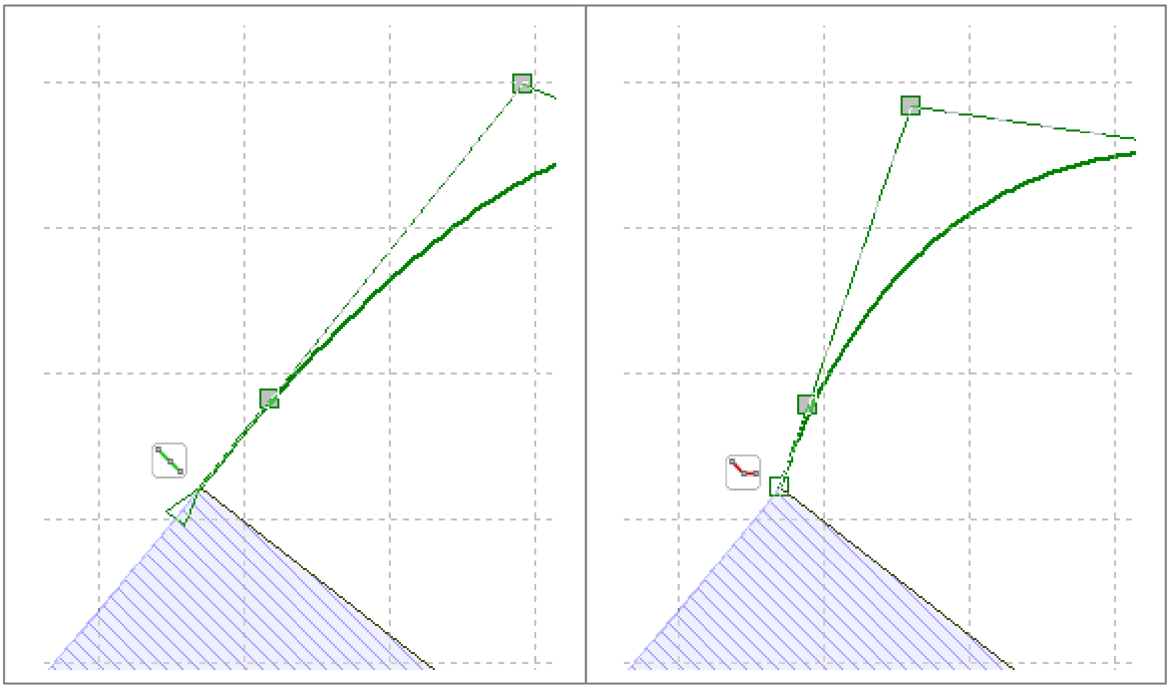
The above mentioned coupling between the Bezier points can be switched on or off by the curve context menu option **Coupled Bezier points**.

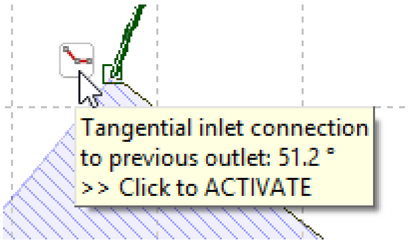
Start angle (line 0-1 or 0-1-2) and end angle (line 3-4 or 2-3-4) can be fixed optionally by the curve context menu option **Fixed start angle** or **Fixed end angle**. A fixed angle is illustrated by a dotted line instead a dashed one and by a triangular marker on the curve endpoint.

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:

tangential connection	non-tangential connection





The hint of the icon contains the angle of the neighboring component for information.

Primary design

For an automatic primary design of the contours the following values are used:

- Main dimensions [190]: d_H, d_S, d_2, b_2
- Inclination angle g of trailing edge to horizontal (see Approximation functions [145])
- Inclination angle e of hub and shroud to vertical (see Approximation functions [145])
- 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) $\Delta z = (d_{1/2} - d_H)/2$

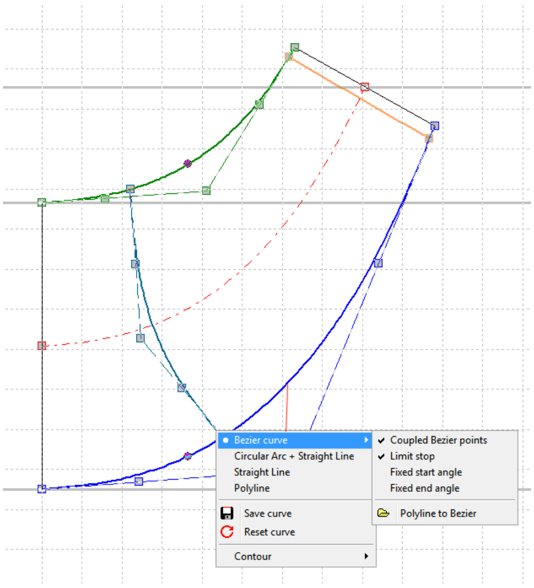
c) $\Delta z = d_2\left(0.014 + 0.023 \cdot \frac{d_2}{d_H} + 1.58 \cdot \varphi\right)$

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.

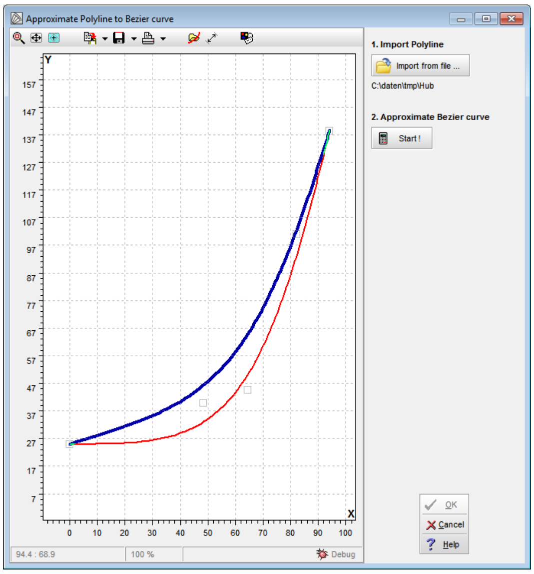
The manipulation of the contours can be achieved by shifting the positions of the Bezier points. As an alternative the position of Bezier points can be realized by input of numerical values (see Graphical dialogs 43). Trailing edge can be rotated by moving Bezier points 4. If <Ctrl> key is pressed simultaneously 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 2πrb should grow from the suction to the impeller diameter as uniformly as possible.

8.2.1.1.1 Converting Polyline / Bezier

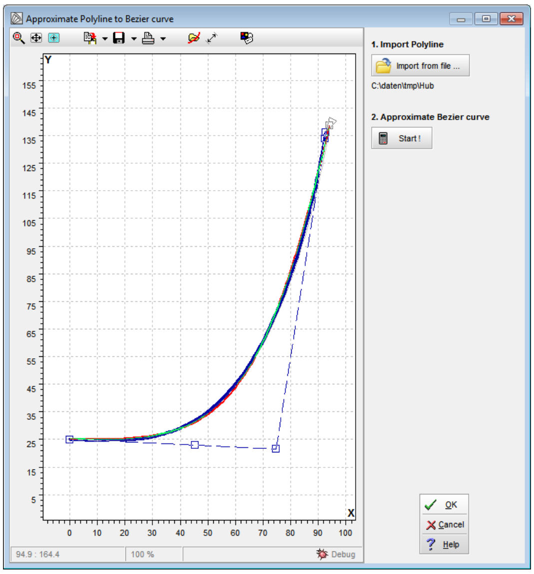


If using simple polyline for hub and/or shroud - e.g. for imported meridional geometrie - this curve can be converted to a Bezier curve. Thus, it's possible to make systematic modifications of existing geometries.



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

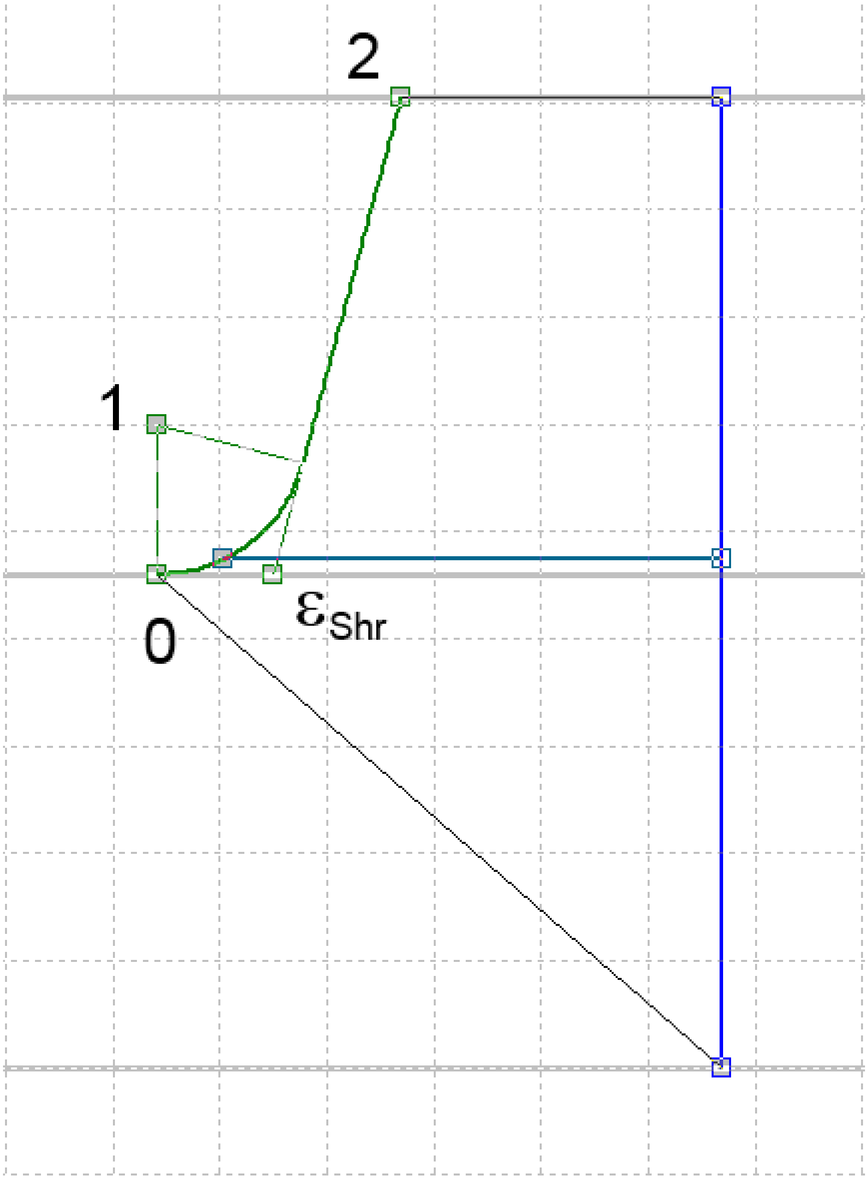
The imported curve is displayed red, the original curve blue.



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.

8.2.1.2 Circular Arc + Straight line

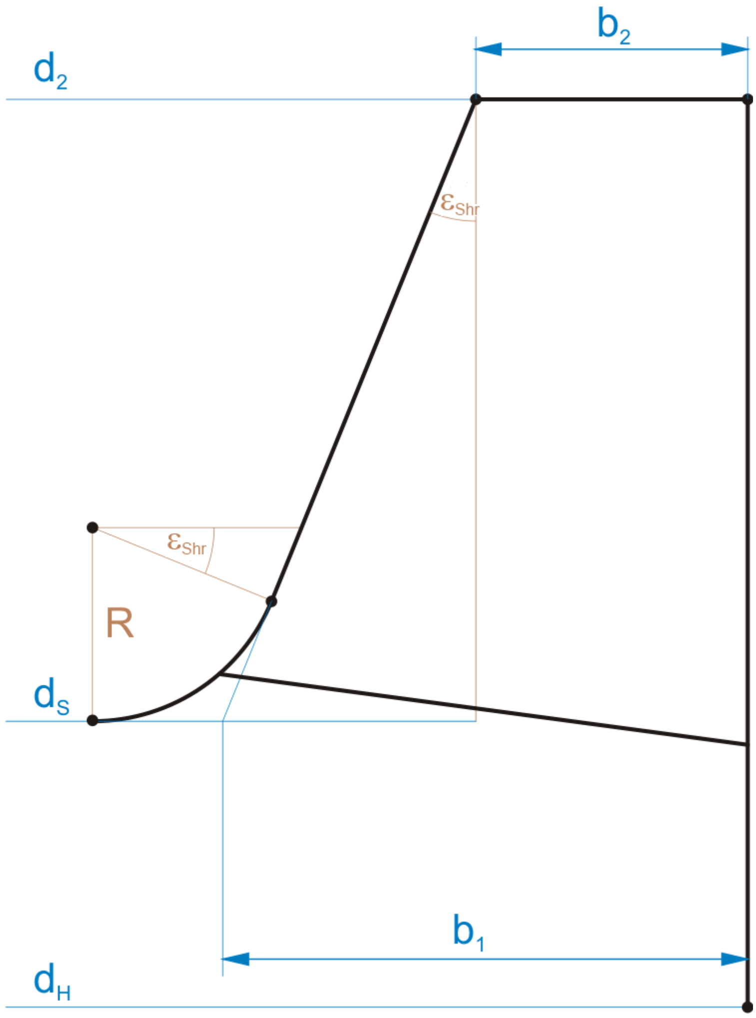
Hub and shroud are represented by the segment of a circle and a tangential straight line. The radius of the segment is defined by Point 1. The points 0 and 2 are defining the axial position of the meridional contour.



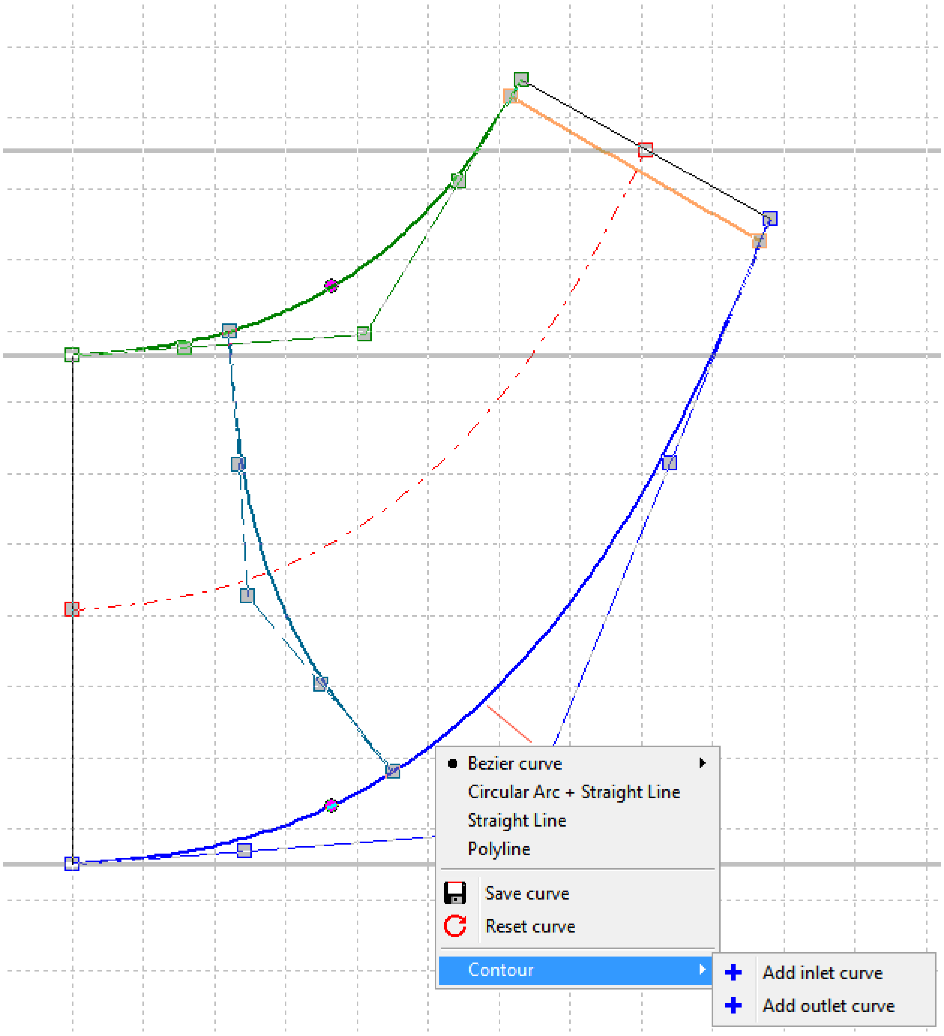
For an automatic primary design of the contours the following values are used:

- Dimensions 190: d_H, d_S, d_2, b_1, b_2
- Radius of the circle segment R: 14% of d_S

The manipulation of the contours can be achieved by shifting the positions of the points. As an alternative the position of points can be realized by input of numerical values. By moving points 0 or 2 the whole geometry can be moved in axial direction.


b2
d2
εShr
εShr
R
dS
b1
dH

8.2.1.3 Contour



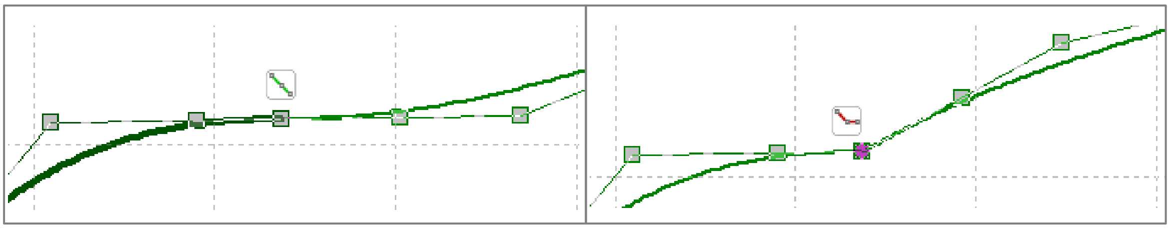
The design of hub and shroud can be expanded optionally. Therefore additional curves can be added on inlet and outlet side in order to design complex contour curves.

The additional inlet and outlet curves can be switched to any curve type (Bezier, Circular, Straight, Polyline) by their own popup menu.

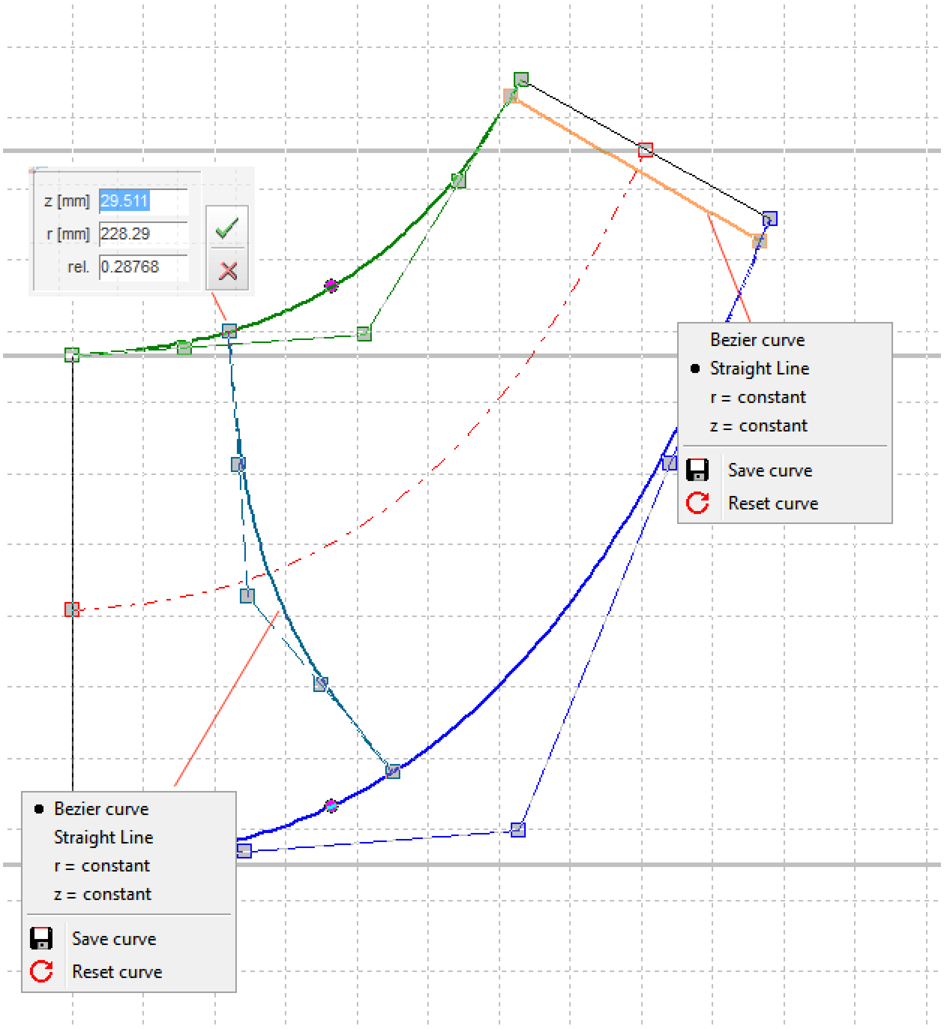
Tangential transition

The tangential transition between neighboring curves can be switched on or off using the icon beside the the first or last Bezier point:

 tangential connection	 non-tangential connection



8.2.2 Leading-Trailing edge contour



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.

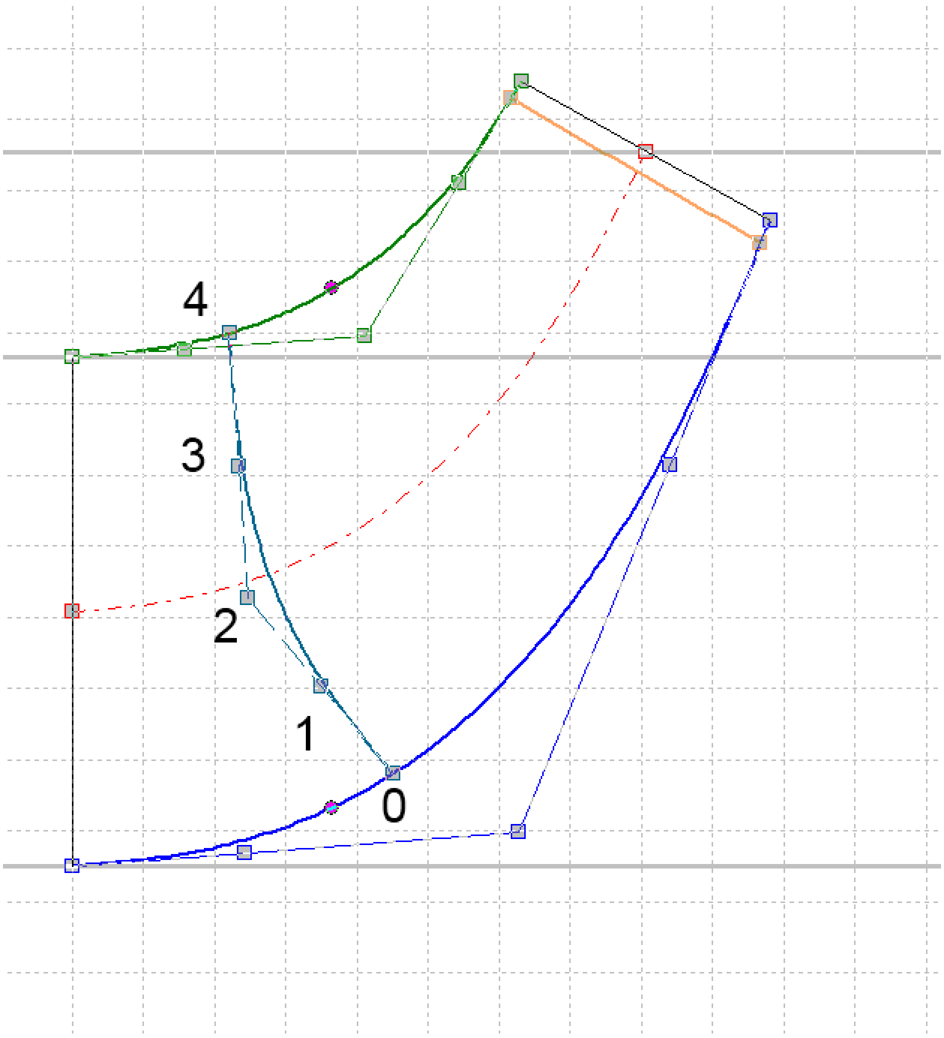
- **z = constant**
  The Leading edge runs on constant axial coordinate, i.e. perpendicular to rotational axis.

The trailing edge can not be designed, if Trailing edge fixed on outlet 270.

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.



The leading edge is designed by a 4th order Bezier curve, too. Regarding the Bezier points, the statements made above are applicable in a similar way. The only difference is the manipulation of the end points. For the leading edge there is no restriction on hub and shroud contour. 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.

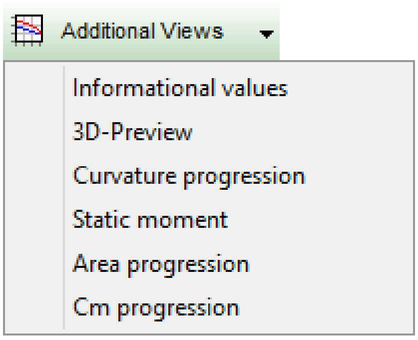
Leading edge can be designed as a straight line by selecting **Straight** in the context menu of the curve (controlled by 2 Bezier points). Additionally the edge can be strictly axial or radial (**z = const.** or **r = const**, controlled by 1 Bezier point).

For radial impellers having nq ≈ 10…30 the leading edge is often designed parallel to the z-axis. As the trailing edge is parallel to the axis too for such applications 2D-curved blades can be created. At higher specific speed 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).

8.2.3 Additional views

The following information can be displayed in the meridional contour dialog using the "Additional views" button:



Informational values

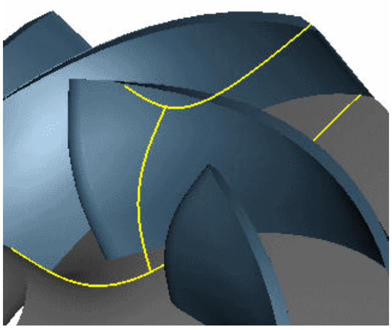
Some additional values are displayed for information:

- 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 ε in the hub and shroud end points measured to the horizontal direction
- Angle γ_{LE} of leading edge on hub and shroud measured to the horizontal direction
- Axial extension Δz of hub and shroud
- Radial extension Δr of hub and shroud
- Angle γ_{TE} of trailing edge measured to the horizontal direction
- Default axial extension Δz_D from inlet shroud to outlet midline (defined for radial impellers only)
- Maximal axial extension Δz_M of complete meridional shape
- Maximal radial extension Δr_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

3D-Preview

3D model 172 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 288

8.2.4 Meridional flow calculation

Stream function ψ

Within the meridian the stream function ψ will be solved. For an incompressible fluid this equation is in cylindrical co-ordinates (z, r):

For a compressible fluid the equation looks like:

where a is the sonic speed defined by:

$$a = \sqrt{\kappa \cdot R \cdot Z \cdot T} \cdot$$

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 setup71.

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 al451.

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:

$$c_r = -\frac{r_R}{r} \frac{\rho_R}{\rho} \frac{\partial \psi}{\partial z},$$

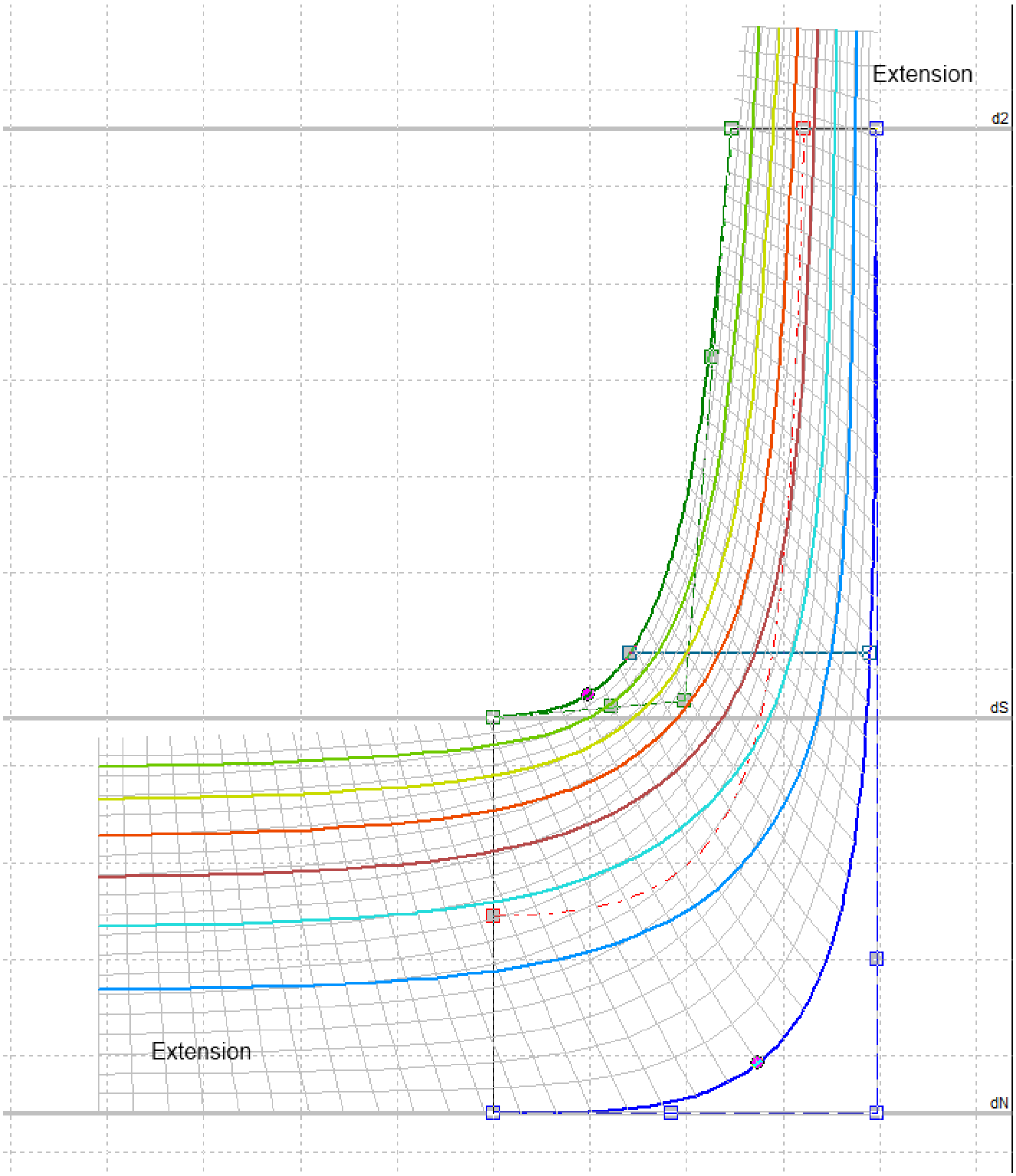
with:

$$c_m = \sqrt{c_z^2 + c_r^2}.$$

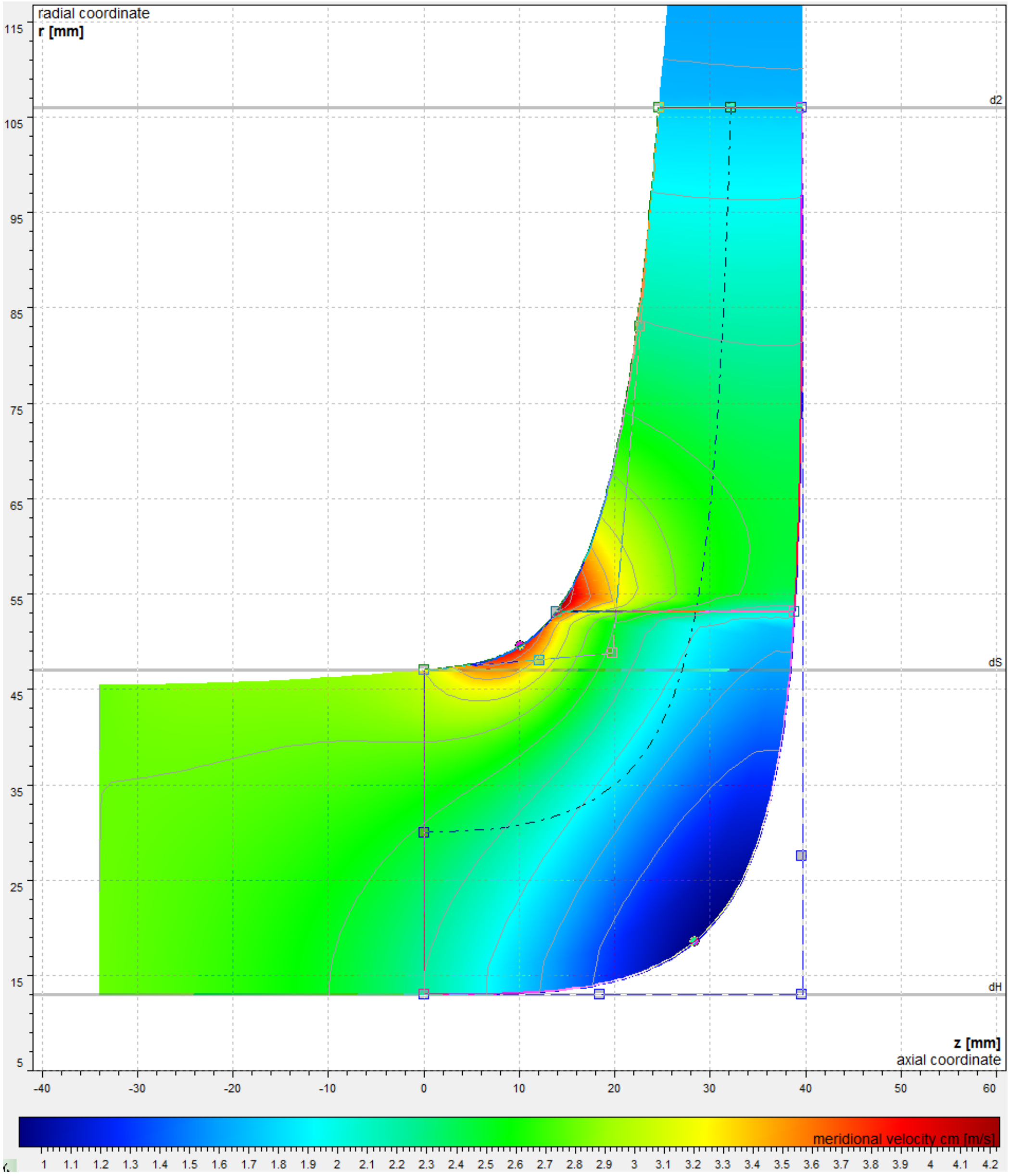
r_R and ρ_R are reference radius and density respectively. In case of incompressible fluids the density is constant throughout the flow domain and the according term in the equations is discarded.

Example

After each change of the meridional contour a new computational grid is calculated. Also, some extensions are added to the inlet and outlet in order to ease the setup of the boundary conditions.



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.

8.3 Mean line design

The design of the blade's geometry is made in four steps in this design mode:

(1) Blade properties 292

(2) Blade mean lines 319

(3) Blade profiles 337

(4) Blade edges 344

8.3.1 Blade properties

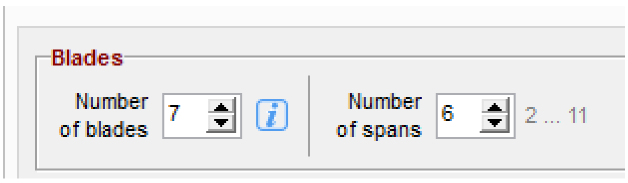
? Impeller | Blade properties

Definition of blade properties is made in two steps:

(1) Blade setup 296

(2) Blade angles 307

Specification of number of blades and number of spans



Usual number of blades are:

Pump	3 … 7
Ventilator	6 … 10
Compressor	Depending on blade exit angle $ß_2$: • 12 for $ß_2$≈30° • 16 for $ß_2$≈45°…60° • 20 for $ß_2$≈70°…90°
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 ... 8.0 for compressors, else 5.0 ... 6.5.

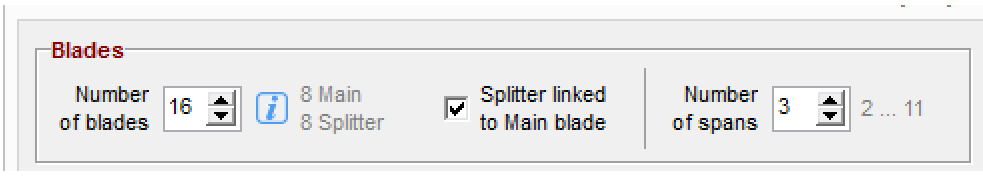
The recommended number of blades using the Zweifel work coefficient is displayed as a hint at the information image **[for axial turbines only]**:

$$z = 2 \cdot \pi \frac{d_{av}}{\psi \cdot \Delta z} \left(\tan(90° - \alpha_1) + \tan(90° - \alpha_2) \right) \cos^2(90° - \alpha_2) ,$$

with Δz the axial chord length and d_{av} the average impeller diameter. The Zweifel work coefficient is in the range of ψ = 0.75..1.15 and is specified in the approximation functions 145.

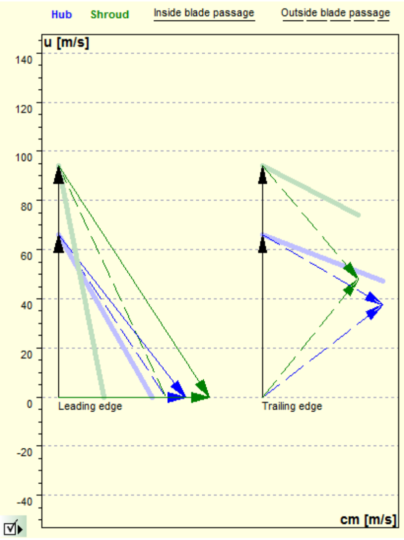
Splitter linked to Main blade

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.



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.

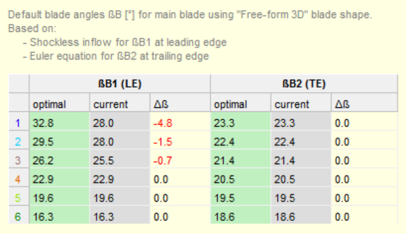
Furthermore the blade angles are displayed by thick lines in order to see the incidence angle on the leading edge and the flow deviation caused by slip velocity on trailing edge.

	Span = 1 (Hub)		Span = 15 (Shroud)	
	Leading edge	Trailing edge	Leading edge	Trailing edge
z	0	55	0	55
d	280	280	399	399
αF	90	52.5	90	38.9
ßF	38.1	59.8	33.1	40.4
u	66	66	94	94
cm	51.7	48.9	61.3	38.9
cu	0	37.5	0	48.2
cr	0	0	0	0
cax	51.7	48.9	61.3	38.9
c	51.7	61.7	61.3	62
wu	66	28.4	94	45.8
w	83.8	56.6	112.2	60.1
τ	1.18	1.09	1.4	1.06
i | δ	-8.2	9.3	-22.2	22.5
w2/w1	0.67		0.54	

(2) Values

Numerical values of velocity components and flow angles are displayed in a table. A short description is at mouse cursor too:

d	Diameter
α	Angle of absolute flow to circumferential direction
β	Angle of relative flow to circumferential direction
u	Circumferential velocity
c_m	Meridional velocity (c_m=w_m)
c_{ax}	Axial component of absolute velocity
c_r	Radial component of absolute velocity
c_u	Circumferential component of absolute velocity
c	Absolute velocity
w_u	Circumferential component of relative velocity: w_u+c_u=
w	Relative velocity
τ	Obstruction by blades (see below)
i	Incidence angle: $i = \beta_{1B} - \beta_1$
δ	Deviation angle: $\delta = \beta_{2B} - \beta_2$
w_R	Deceleration ratio of relative velocity: $w_R=w_2/w_1$



Default blade angles ßB [°] for main blade using "Free-form 3D" blade shape.
Based on:
- Shockless inflow for ßB1 at leading edge
- Euler equation for ßB2 at trailing edge

	ßB1 (LE)			ßB2 (TE)		
	optimal	current	Δß	optimal	current	Δß
1	32.8	28.0	-4.8	23.3	23.3	0.0
2	29.5	28.0	-1.5	22.4	22.4	0.0
3	26.2	25.5	-0.7	21.4	21.4	0.0
4	22.9	22.9	0.0	20.5	20.5	0.0
5	19.6	19.6	0.0	19.5	19.5	0.0
6	16.3	16.3	0.0	18.6	18.6	0.0

(3) Default ßB, mean line design only

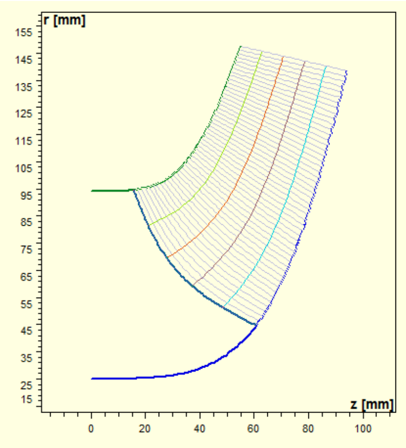
Default blade angles for the optimal Free-form 3D blade shape is displayed compared to the currently specified/ calculated angles. Deviations from default values are marked in red color. Default blade angles are calculated based on

- Shockless inflow for $ß_{B1}$ at blade leading edge
- Euler equation for $ß_{B2}$ 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" 309.

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

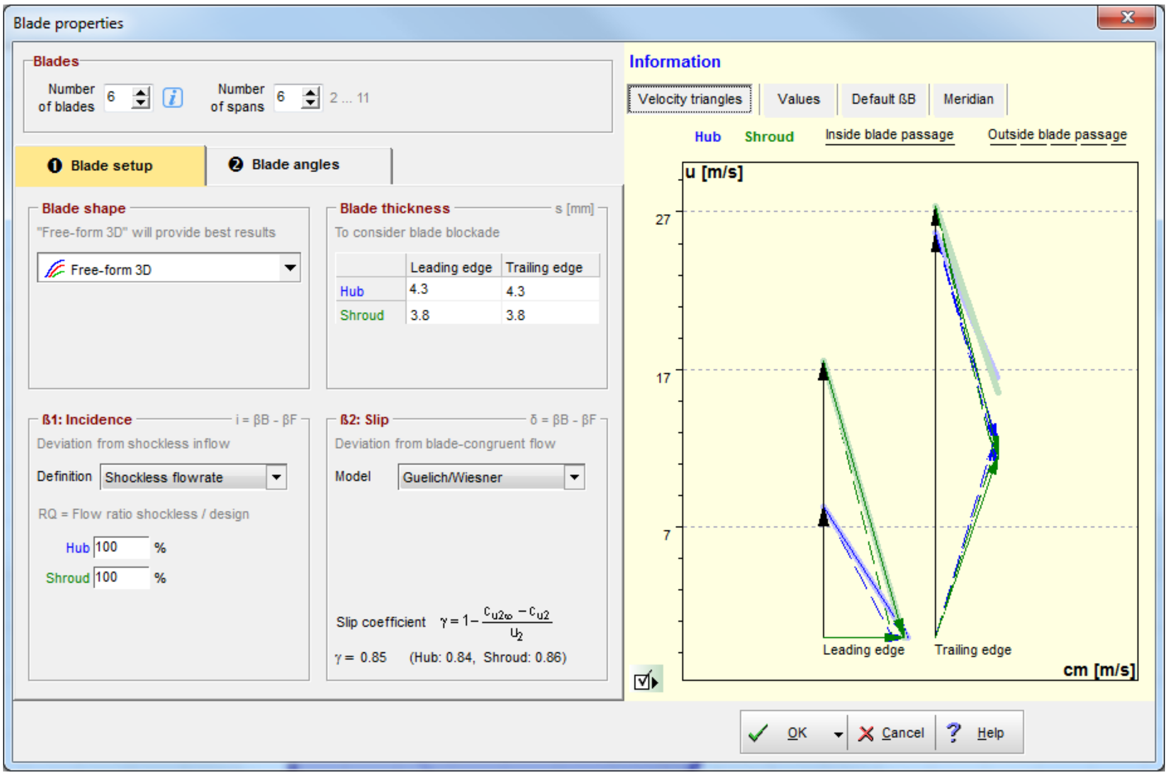
For Radial element blades the number of spans is fixed to 11. Furthermore a **Distribution exponent** can be specified. This exponent has influence on the distribution of spans and herewith especially on the shape of the leading edge (turbine). For highly spatial curved blades the continuity of the blade surface can be influenced by this parameter.

Distribution exponent	Impact
Distribution exponent 1	1: spans uniformly distribut (default)
Distribution exponent 0.5	<1: spans concentrated tov shroud
Distribution exponent 1.5	>1: spans concentrated tov hub

8.3.1.1 Blade setup

? Impeller | Blade properties

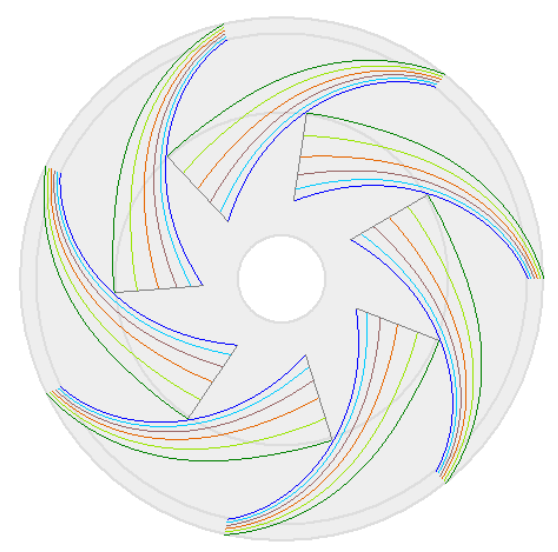
On page **Blade setup** basic blade properties are defined.



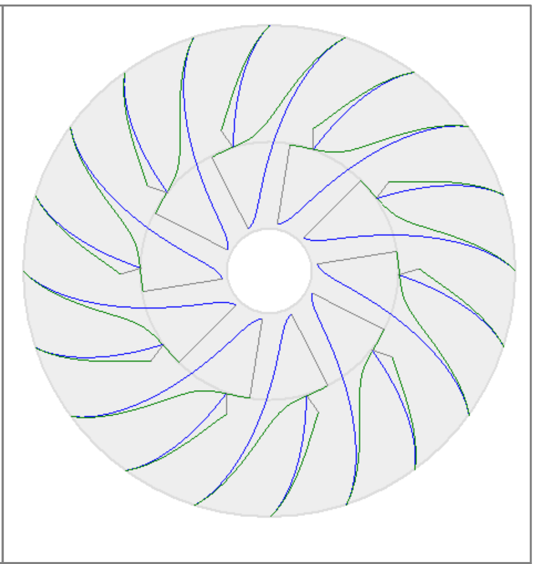
(1) Selection of desired Blade shape

There are 7 different types:

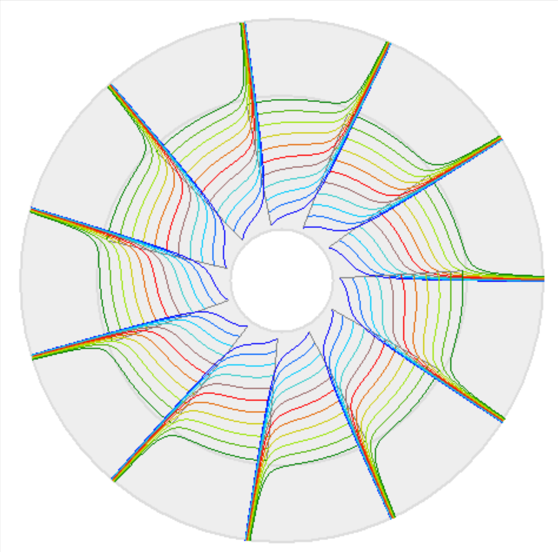
Free-form 3D	Ruled surface 3D 304



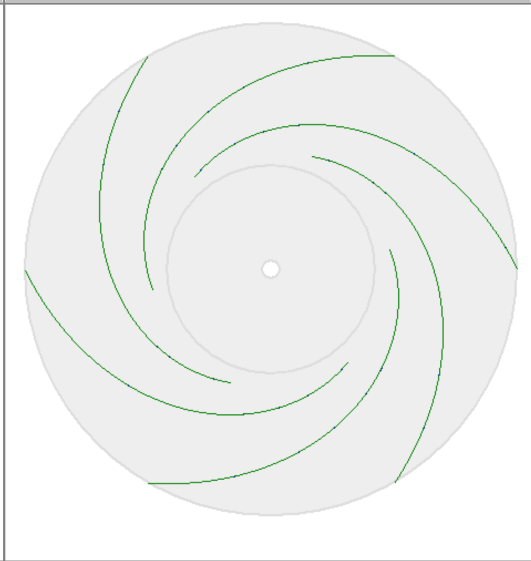
[Radial elements 3D](306)



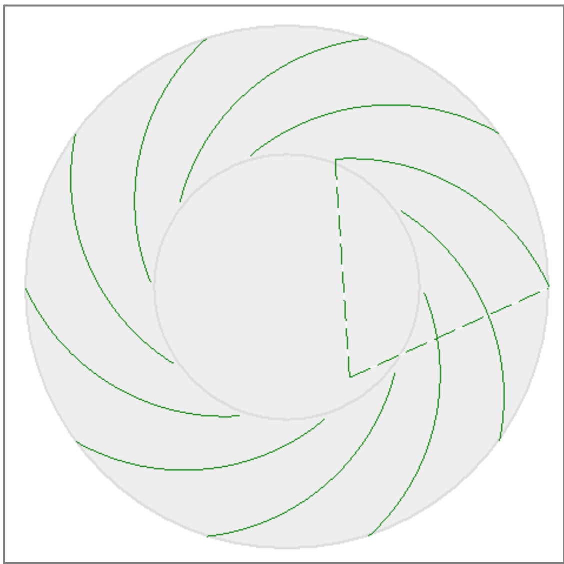
Free-form 2D



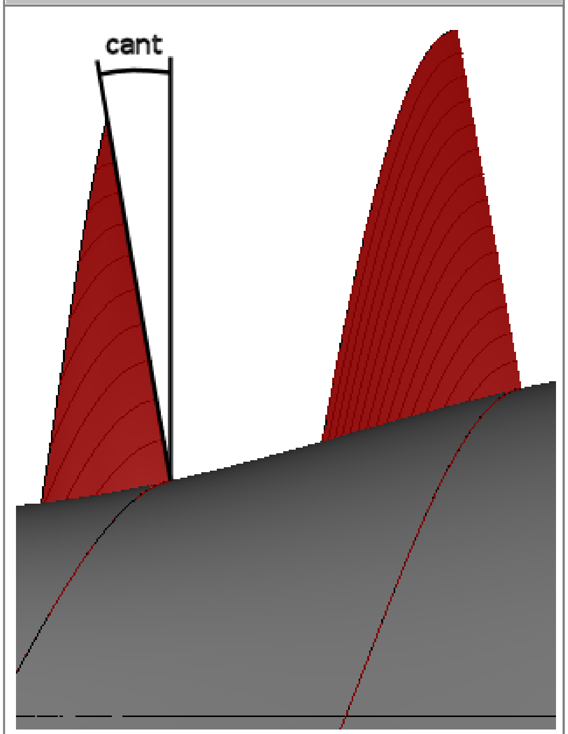
Circular 2D



Straight 2D



Helical 3D



The initial blade shape depends on the machine type and can be customized in the Impeller preferences 161.

PUMP	
Radial & Mixed-flow	Free-form 3D

+ Waste water pump	Free-form 2D
Axial	Free-form 3D
+ Inducer	Helical 3D
VENTILATOR	
Radial & Mixed-flow	Circular 2D
Axial	Free-form 3D
COMPRESSOR	
Radial & Mixed-flow	Ruled surface 3D
TURBINE	
Radial & Mixed-flow	Radial elements 3D 306
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 304.

Limitations

Blade shape	Splitter blades	Meridional shape
Free-form 3D	(no limitations)	
Ruled surface 3D		
Radial elements 3D		
Helical 3D	for Inducers only	
Free-form 2D		available only if the meridional direction is mainly radial: hub must overlap shroud in z-direction about 50% or more
Circular 2D	not available for splitter blades	
Straight 2D		

(2) Defining the blade thickness values at leading and trailing edge in panel Blade thickness s

Blade thickness is 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 145.

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.

(3) Specification of incidence angle on blade leading edge (deviation from shockless inflow) on panel ß1: Incidence

Pump, Ventilator, Compressor		Turbine
from ratio Q for shockless inflow / Q for max. efficiency	$R_Q = Q_{shockless} / Q_{BE}$	fully automatic by theory of **WIESNER** adapted by Aungier 312
or directly by incidence angle i (RQ=100% or i=0° for shockless inflow)		*or* directly by incidence angle i (i=0° for shockless inflow)
or from ratio of incidence angle i / blade angle β_B	$i_{Rel} = i / \beta_B$	

For inducers there is an additional check if the incidence is > 1° even for high flow rates (overload) to prevent pressure side cavitation.

[Pump, Ventilator, Compressor impellers only]

(4) Estimation of slip velocity in panel β2: Deviation flow – blade

You have to use one of the following slip models:

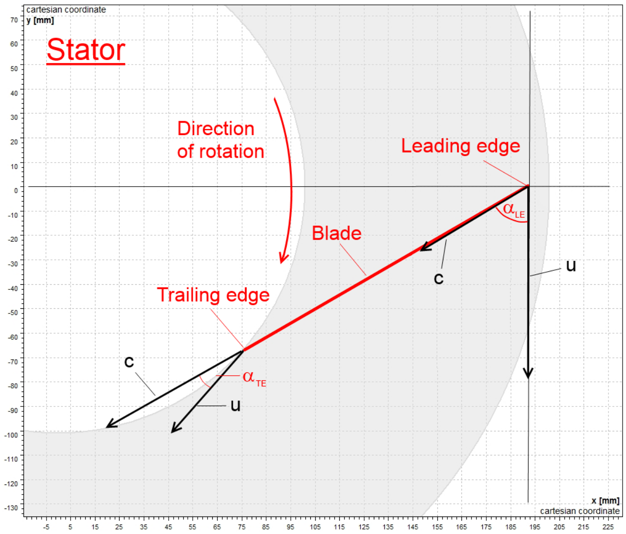
- **WIESNER** 318 theory
  closed empirical model
- **AUNGIER** 316 theory
  closed empirical model, extended Wiesner model
- **PFLEIDERER** 317 theory
  input of coefficient a
- **User-defined**
  manual selection of angular deviation $ß_{2B}$-$ß_{2}$ resp. velocity ratio $c_{u2}/c_{u2,\infty}$
- **GUELICH** 319 theory (for waste water pumps only)
  specific slip model for waste water pump design

the radius of leading edge varies from hub to shroud the blade angle β_{1B} does not remain constant. A higher radius on shroud results in a lower value for β_{1B}- the blade is curved on leading edge.

Possible warnings

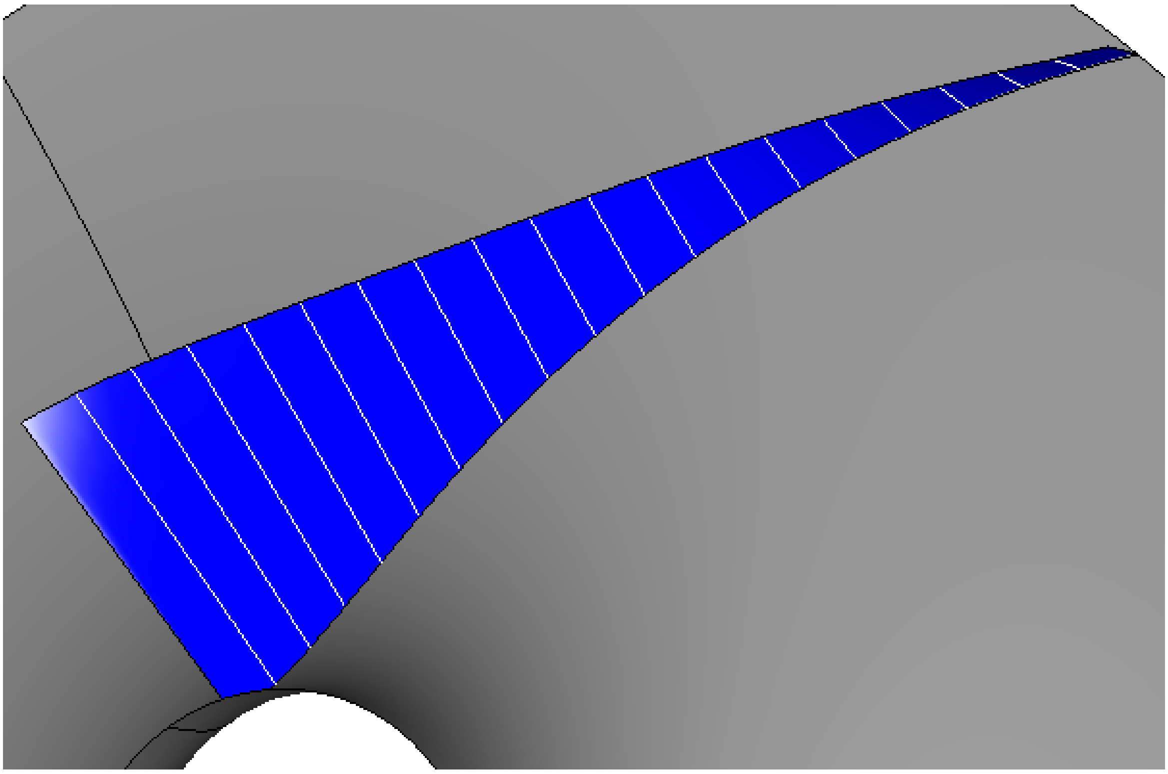
Problem	Possible solutions
Number of blades differ from the initially defined (Main Dimensions). (waste water pumps only)	
Number of blades differs from the number that was initially selected in Main dimensions 190 used for empirical correlations to calculate the main dimensions. This can result in inconsistent impeller design.	It makes no sense to use other number of blades for main dimension calculation and blade design itself. Before modifying the number of blades here one should adapt the number in Main dimensions 190, update the empirical parameters and the main dimension.
All mean lines except the hub mean line are extrapolated on the leading/ trailing edge. ("Free-form 2D" blade shape only)	
The hub is the master mean line for "Free-form 2D" blade shape. For this blade shape the geometry of all	Use axis parallel (const. radius) or slightly sloping meridional leading/ trailing edge.

Problem	Possible solutions
other mean lines is designed automatically in such way that it is exactly overlapping the hub mean line if viewing in z-direction. The resulting blade shape is two-dimensional. If the other curves have points with higher radius at trailing edge/ lower radius at leading edge than the last/ first hub point (sloping meridional edge), then these curves have to be extrapolated.	**Leading edge**: The shroud point should have higher or equal radius than the hub point. **Trailing edge**: The shroud point should have lower or equal radius than the hub point.
"Radial elements 3D" blade shape is not possible for the current combination of meridional leading/trailing edge and hub contour	
The hub is the master mean line for "Radial elements 3D" blade shape. The geometry of all other mean lines is designed automatically in such way that it forms a blade consisting of radial fibers 306. The resulting blade shape is three-dimensional. If the other curves have points with lower z-values at leading edge/ higher z-value at trailing edge than the first/last hub point, these curves have to be extrapolated. In this case the blade would have a bad quality in the extrapolated region.	Use radial (const. axial position) or sloping meridional leading/ trailing edge. **Leading edge**: The shroud leading edge should have a higher or equal axial position compared to the hub. **Trailing edge**: The shroud trailing edge should have a lower or equal axial position compared to the hub.
Ruledsurface blades may have bad quality surfaces in case of just 2 mean lines ("Ruled surface 3D" blade shape only)	
Impeller with splitter blades can have wavy blade surface if only 2 blade profile sections are used.	Increase the number of blade profile sections (page "Blade angles").
"Straight 2D" blade shape is not possible for the current meridional leading edge contour and blade angle combination.	
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. If the other mean lines are extended they will be extrapolated automatically. For specific combinations of meridional leading edge and blade angles **βB1** 307 an extrapolation is impossible.	**Leading edge** 284: The point on shroud should be moved to a higher radius. **βB1** 307: Blade angle should be increased.
"Straight 2D" blade shape is not possible for the current meridional trailing edge contour and blade angle combination.	

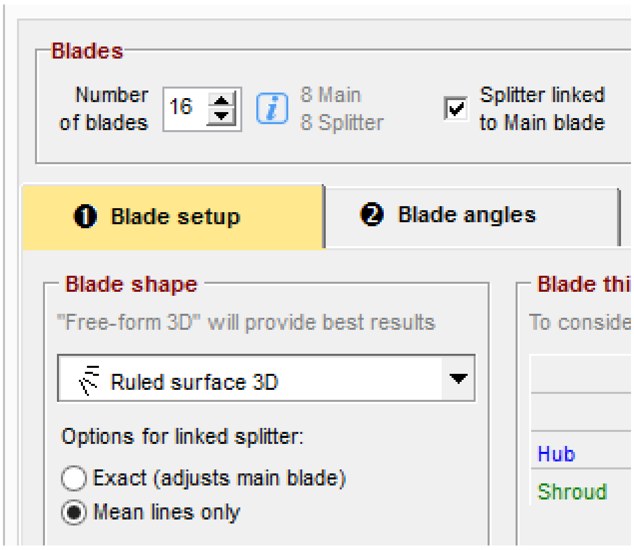
Problem	Possible solutions
The blade angle is too small or too large - therefore designing a "Straight 2D" blade shape is impossible.  	**Trailing edge** 284: The edge should be moved to a higher radius. **αLE/βLE** 307: Blade angle should be increased.

8.3.1.1.1 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.

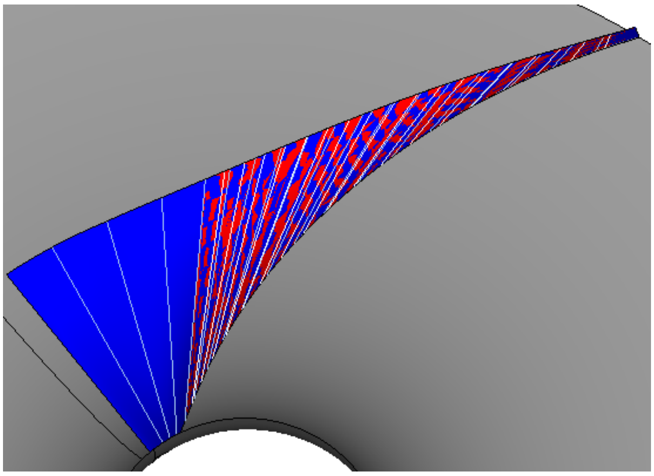


You can choose between the following options:

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.

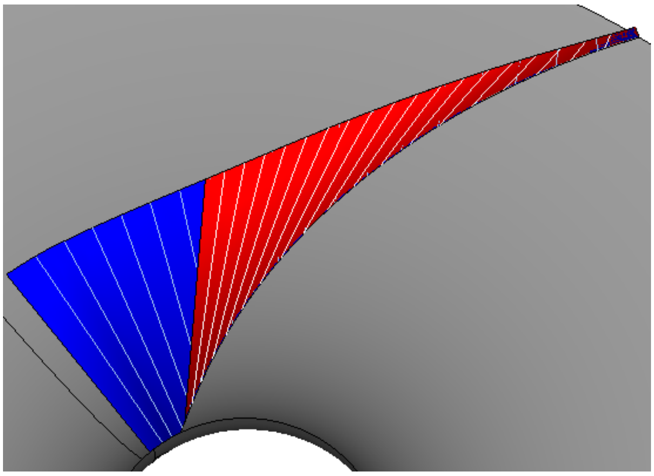
Mean lines only: The blade geometry of the splitter is designed using the mean lines of the main blade. The advantage of this option is a higher flexibility in design of a curved leading edge of the splitter. (depends on the number of mean lines)

The following pictures illustrate the combination of different options (splitter is rotated into the main blade for illustration):



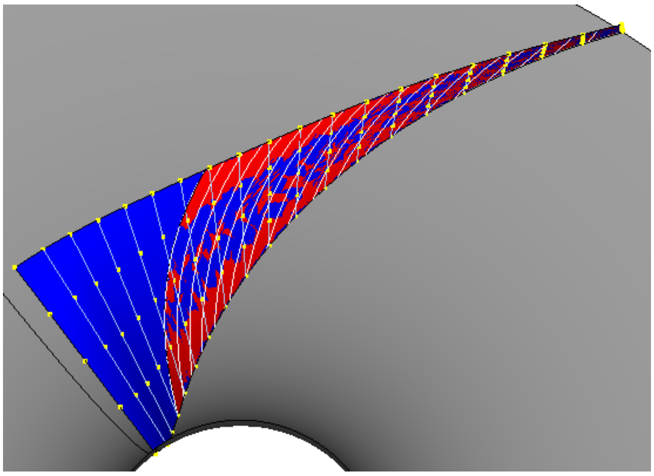
- Splitter linked to Main Blade
- 2 spans
- Exact (adjusts main blade)

Main and Splitter are using identic rulings. The splitter leading edge is influencing the rulings and therefore the main blade.



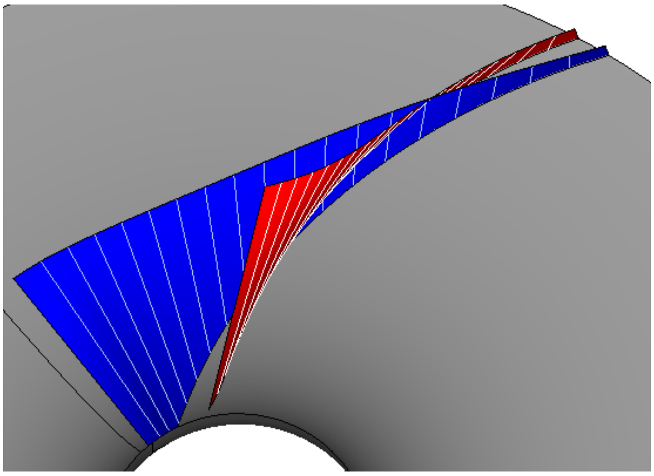
- Splitter linked to Main Blade
- 2 spans
- Mean lines only

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

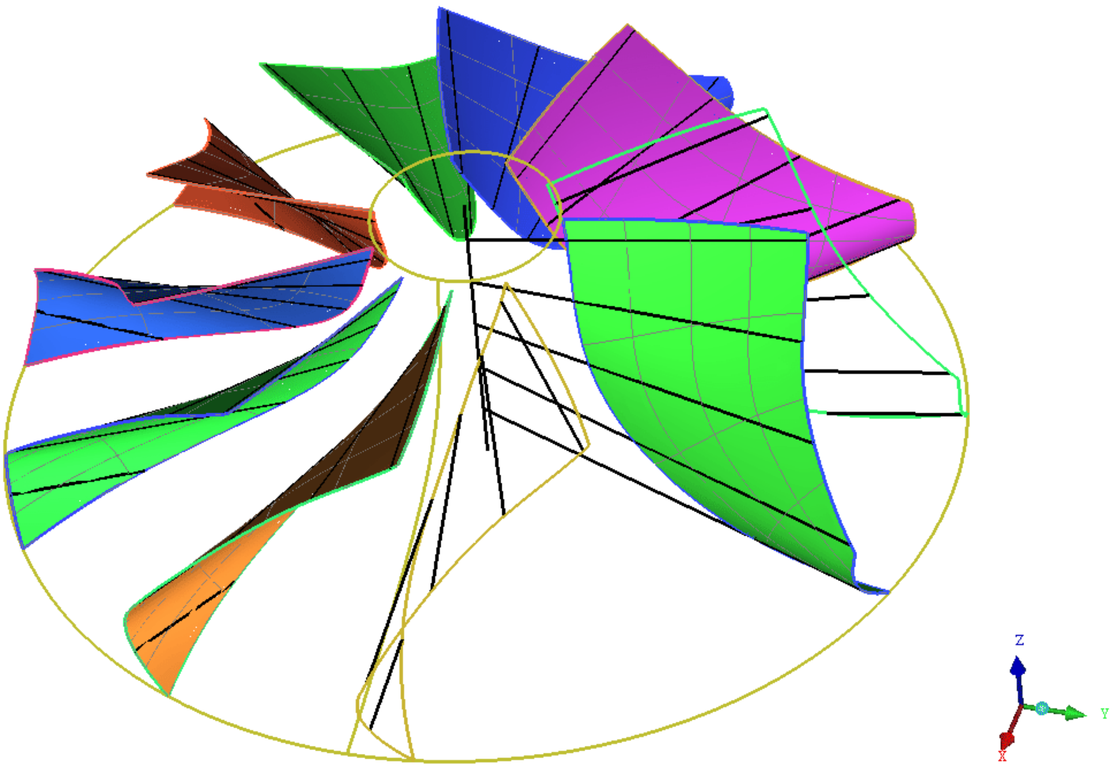
Main and splitter are using their own rulings. There is no coupling between splitter and main blade. The splitter shape can highly deviate from the main blade.

8.3.1.1.2 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 = constant.

Radial element blades require the following geometrical boundary conditions for radial & mixed-flow impellers:

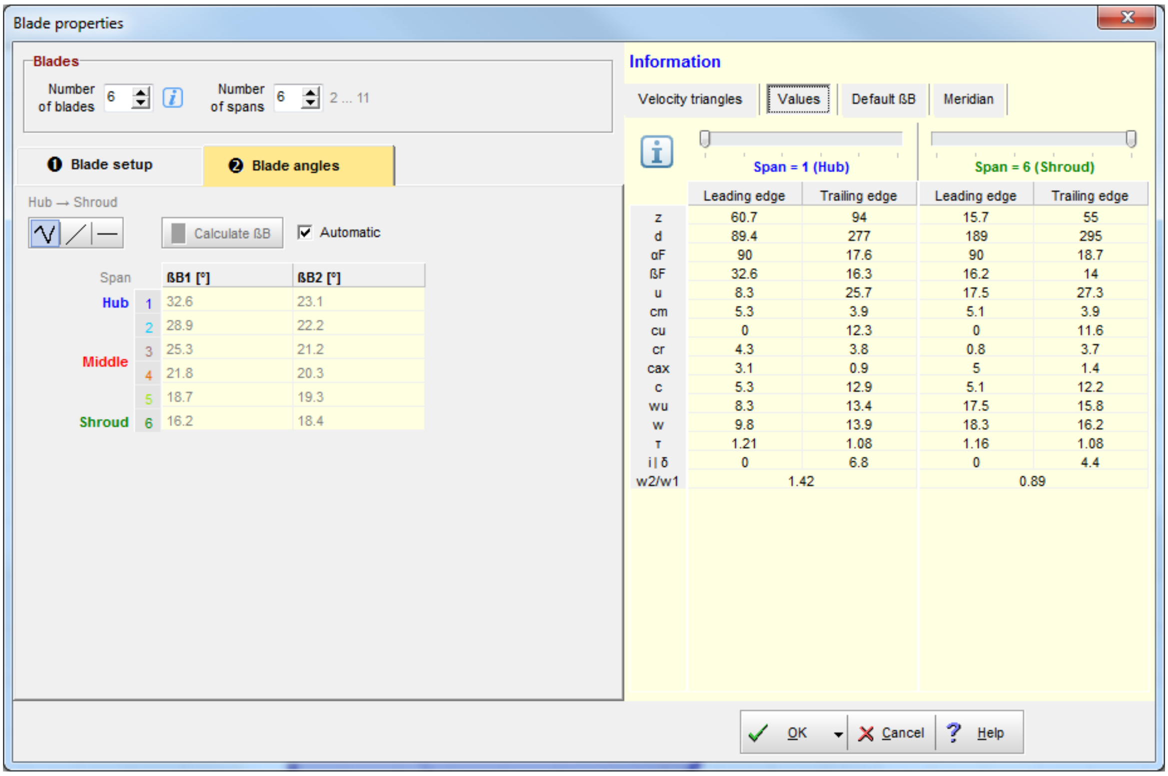
- Blade angle at input (turbines) or output resp. (all other types): $\beta \approx 90°$
- Inclination angle 276 from hub and shroud to the horizontal: $\varepsilon' < 90°$
- Vertical trailing (turbines) or leading edge resp. (all other types) with $z \approx$ const.
- Small wrap angle: $\varphi \approx 360°$/number of blades



8.3.1.2 Blade angles

? Impeller | Blade properties

On this page the **blade angles** are calculated.



Later designed mean lines depend on the number and the meridional position of profile sections as well as the blade angles. Blade angles 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 319 calculation.

Distribution from hub to shroud

The blade angles are calculated for hub and shroud. On panel **Distribution from hub to shroud** you can define how the blade angles of the inner sections are defined.

Blade angles βB

- Specifying number of blade profile sections for further blade design using the vertical track bar
- Calculation of blade angles using values from Blade setup 296 by pressing button **Calculate βB**
- Manual adaptation of calculated blade angles if required

Calculation or input of blade angles can be executed for 2 up to 11 blade profiles. Further blade design is realized according to the defined blade profile number. All meridional lines which will be used for blade design are displayed in the diagram. The number of lines can be adjusted with the track bar on the left side of the table. By default the meridional lines are equally spaced between hub and shroud.

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 βB**. This recalculation is made automatically if the checkbox **Automatic** is selected.

➜ Details of calculation of Inlet triangle 310

➜ Details of calculation of Outlet triangle 313

(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 319 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" 328 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

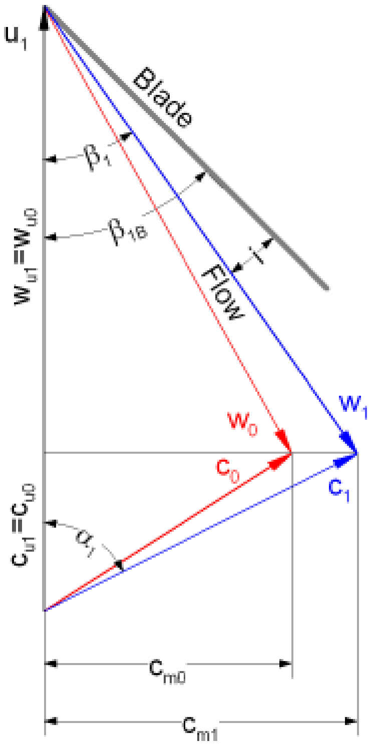
Problem	Possible solutions
Blade angles are updated automatically. Therefore geometry modifications are possible.	
Blade angles are updated automatically if any input parameters are modified.	To fix the blade angles you could uncheck the "Automatic" calculation. Then you have to

Problem	Possible solutions
	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.
Change of swirl cu*r is wrong. Check blade angles and velocity triangles.	
$c_{u2}*r_2$ is lower than $c_{u1}*r_1$ (turbines: $c_{u2}*r_2$ is higher than $c_{u1}*r_1$) resulting in energy transmission in the wrong direction (Euler equation of turbomachinery).	Recalculate and/or check blade angles β_B and flow angles β at leading and trailing edge.
ΔβB1/2 (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.
ΔβB1/2 (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.

8.3.1.2.1 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 190) 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.



$$\tan\beta_1 = \frac{c_{m1}}{w_{u1}}$$

$$c_{m1} = c_{m0}\ \tau_1$$

$$\tau_1 = \frac{t_1}{t_1 - \sigma_1} \quad \text{with} \quad t_1 = \frac{\pi d_1}{z}, \sigma_1 = \frac{s_1}{\sin\beta_{1B}}$$

$$c_{m0} = Q/(\pi d_1 b_1)$$

$$w_{u1} = u_1 - c_{u1}$$

$$u_1 = \pi d_1 n$$

$$c_{u1} = c_{uS}\frac{r_S}{r_1} = u_S(1-\delta_r)\frac{r_S}{r_1} \quad \text{(const. inflow swirl)}$$

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 = β_{1B}-β_1

In general an inflow without any incidence is intended (i=0). If i≠0 the flow around the leading edge shows high local velocities and low static pressure:

i > 0: $\beta_1 < \beta_{1B} \rightarrow$ stagnation point on pressure side
i < 0: $\beta_1 > \beta_{1B} \rightarrow$ stagnation point on suction side

A small incidence angle i can be profitable for best efficiency point. Calculation of β_{1B} inside CFturbo gives inflow without incidence.

Typical inlet blade angles are:

Pumps, Ventilators	β_{1B} < 40° due to best efficiency
Pumps	β_{1B} as small as possible due to cavitation; with regard to efficiency not smaller then 15…18°
Compressors	optimal blade angle β_{1B} is about 30°

If the radius of leading edge varies from hub to shroud the blade angle β_{1B} does not remain constant. A higher radius on shroud results in a lower value for β_{1B}- the blade is curved on leading edge.

Possible warnings

Problem	Possible solutions
Leading edge blade angle β1 > 40° (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 suction diameter d_S (Main dimensions 190)
Leading edge blade angle ß1 < 10°	
Unusual low inlet blade angles.	Too small inlet angles indicate too high inlet cross section. Decrease suction diameter d_S (Main dimensions 190)
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.
ßB indeterminate. It's not possible to determine blade angle ßB.	
Blade angle calculation failed.	Check input values and geometry.

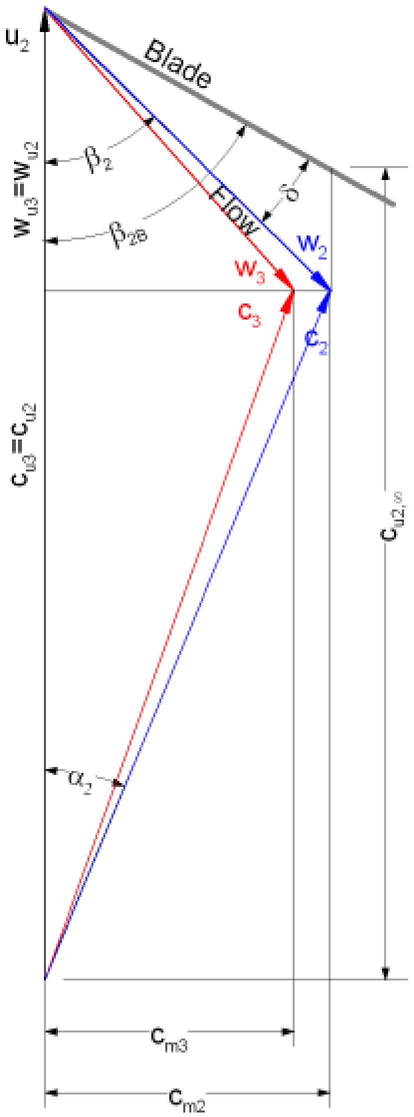
[Turbine rotors only]

In case of **turbines** the calculation of the incidence by Aungier 316 can be used.

According to decreased energy transmission the slip coefficient γ is defined:

8.3.1.2.2 Outlet triangle

The outlet triangle is determined by geometrical dimensions of flow channel and selected blade angle β_{2B}. The blade angle β_{2B} 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.

$$\tan\beta_2 = \frac{c_{m2}}{w_{u2}}$$

$$c_{m2} = c_{m3}\tau_2$$

$$\tau_2 = \frac{t_2}{t_2 - \sigma_2} \quad \text{with} \quad t_2 = \frac{\pi d_2}{z}, \sigma_2 = \frac{s_2}{\sin\beta_{2B}}$$

$$c_{m3} = Q/(\pi d_2 b_2)$$

$$w_{u2} = u_2 - c_{u2}$$

$$u_2 = \pi d_2 n$$

$$c_{u2} = \frac{Y/\eta_h + u_1{}^2(1-\delta_r)}{u_2}$$

$$\text{from}: \tilde{Y} = \frac{Y}{\eta_h} = u_2 c_{u3} - u_1 c_{u0}$$

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 δ:

$\delta = \beta_{2B} - \beta_2$

The deviation angle should not exceed 10°…14°, 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_{2B} 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. Three different possibilities are available in CFturbo (not for **Turbines**):

→ (1) Decreased output by PFLEIDERER 317

→ (2) Outflow coefficient by WIESNER 318

→ (3) Outflow coefficient by AUNGIER 316

Blade angle β_{2B} must be determined to reach the desired energy transmission - respectively the required head/ pressure difference - under consideration of slip velocity.

The following recommendations for common blade angles β_{2B} 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 331

Radial machines - except for turbines - with low specific speed nq usually have similar values for β_{2B}. The blades for this type of impellers are often designed with a straight trailing edge (β_{2B}=const.). For turbine rotors the radii along the trailing edge from hub to shroud are very different, resulting in very different values for β_{2B} and twisted blades.

Possible warnings

Problem	Possible solutions
Trailing edge blade angle ß2 < 10°	
Unusual low outlet blade angles	Too small outlet angles indicate too high outlet

Problem	Possible solutions
	cross section. Decrease impeller diameter d_2 or outlet width b_2 (Main dimensions 190)
The deviation (slip) between blade and flow δ > 20° (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 190), increase the number of blades, increase meridional blade length (Meridional contour 268), select a different slip model
Trailing edge blade angle ßB2 > 90°.	
Unusual high blade angles at trailing edge. The blades are forward curved.	Increase impeller diameter d_2 or outlet width b_2 (Main dimensions 190) and/or the slip coefficient γ.
The 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.
ßB indeterminate. It's not possible to determine blade angle ßB.	
Blade angle calculation failed.	Try to increase the impeller diameter d_2 or outlet width b_2 and/or the slip coefficient γ.
The deviation (slip) between blade and flow δ is unrealistic high. Check deviation model and/or values.	
The slip calculation results in a value higher than 90°, which is unrealistic high.	Possible solutions could be: increase the impeller diameter (Main dimensions 190), increase the number of blades, increase meridional blade length (Meridional contour 268), select a different slip model

8.3.1.2.2.1 Slip coefficient by AUNGIER

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 318 original empirical equation for the estimation of outflow coefficient:

$$\gamma = 1 - \frac{\sqrt{\sin \beta_{2B}}}{z^{0.7}}$$

The limiting radius ratio ε_{Lim} is given by:

$$\varepsilon_{Lim} = \frac{\gamma - \sin(19° + 0.2\beta_{2B})}{1 - \sin(19° + 0.2\beta_{2B})}$$

The slip factor is corrected for radius ratios $\varepsilon = r/r_2 > \varepsilon_{Lim}$ with:

$$\gamma_{cor} = \gamma \left(1 - \left(\frac{\varepsilon - \varepsilon_{Lim}}{1 - \varepsilon_{Lim}} \right)^{\sqrt{\beta_{2B/10}}} \right)$$

[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.

Circumferential component of blade congruent flow can be calculated as follows:

8.3.1.2.2.2 Slip coefficient by PFLEIDERER

Reduced energy transmission is expressed by decreased output coefficient p:

$$p = \frac{\tilde{Y}_\infty}{\tilde{Y}} - 1$$

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

$$\psi' = a\left(1 + \beta_2 / 60^\circ\right)$$ experience number

experience number a:

Radial impeller	with guided vanes	a = 0.6
	with volute	a = 0.65…0.85
	with plain diffusor	a = 0.85…1.0
Mixed flow/axial impeller		a = 1.0 …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=1: for flow congruent to blade)

Circumferential component of the flow, which is congruent to blade, can be calculated as follows:

$$c_{u2\infty} = \frac{c_{u2}}{k_L} - \frac{r_1^{\,2}}{r_2}\left(\frac{1}{k_L} - 1\right)2\pi n\left(1 - \delta_r\right)$$

Now the outflow (slip) coefficient γ according to Wiesner 318 can be calculated:

$$\gamma = 1 - \frac{c_{u2\infty} - c_{u2}}{u_2}$$

8.3.1.2.2.3 Slip coefficient by 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.

Wiesner developed an empirical equation for the estimation of outflow coefficient:

$$\gamma = 1 - \frac{\sqrt{\sin\beta_{2B}}}{z^{0.7}}$$

Gülich modified this formula by two additional correction factors:

$$\gamma = f_1\left(1 - \frac{\sqrt{\sin\beta_{2B}}}{z^{0.7}}\right)k_w$$

with the correction factors:

$$\varepsilon_{Lim} = \exp\left(-\frac{8.16 \sin\beta_{2B}}{z}\right)$$

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_{1m} is not used for the calculation of k_w in CFturbo but the diameter at hub and shroud respectively. Doing so a slip coefficient for hub and shroud can be calculated. An average slip coefficient is determined by:

$$\gamma = 0.5(\gamma_{Hub} + \gamma_{Shroud})$$

The switch between radial and mixed-flow calculation of the correction factor f_1 is done by:

$$f_1 = \max\left(0.98, 1.02 + 1.2\cdot 10^{-3}\left(n_q - 50\right)\right)$$

8.3.1.2.2.4 Slip coefficient by GÜLICH (waste water pumps)

For waste water pumps the slip mainly depends on the number of blades.

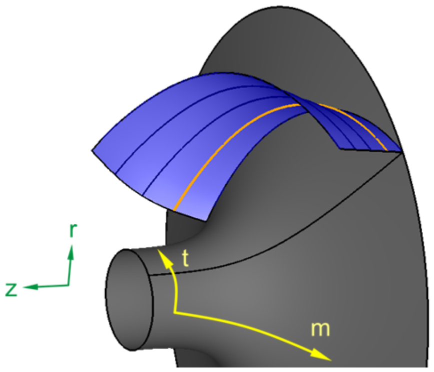
The table contains typical values for the slip coefficient γ:

number of blades	slip coefficient γ
1	0.48 … 0.6
2	0.53 … 0.65
3	0.67 … 0.75

8.3.2 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 292.



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:

$$dm = \frac{dM}{r} \qquad dt = \frac{dT}{r}$$

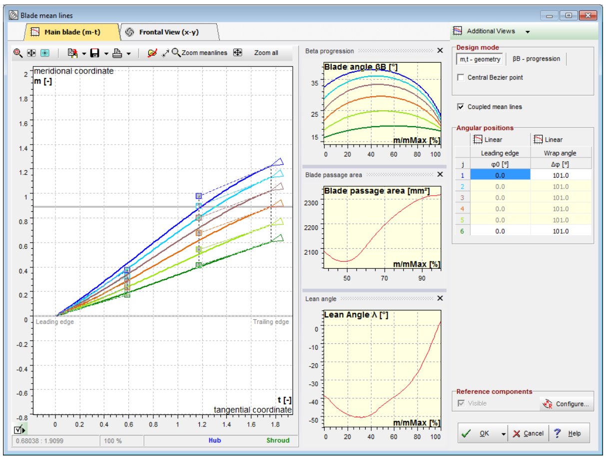
$$\tan\beta = \frac{dm}{dt}$$

This conformal mapping allows the uniform handling of various impeller types (radial, mixed-flow, axial).

It should be noted that for each meridional flow surface a separate m-coordinate is existing.

Design mode

The mean lines can be designed on 2 alternative methods. On panel Design mode you can select:

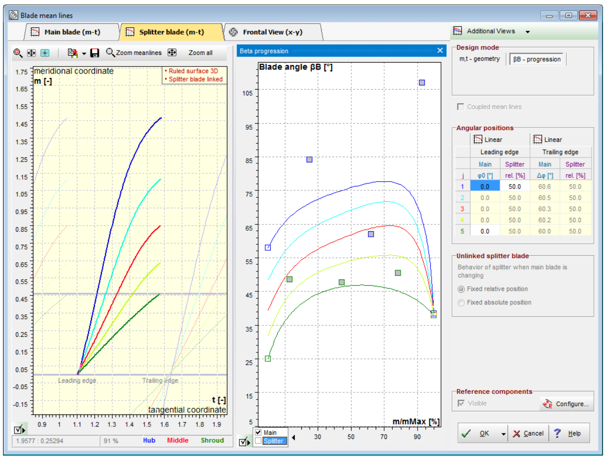


m,t-geometry

The blade is designed in the conformal m,t-mapping by Bezier curves. Beta distribution as an additional view is calculated and displayed for checking.

 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.

Also the visibility of the inner mean lines can be toggled via "**Inner mean lines**".



βB progression

The blade is designed via its Beta distribution by Bezier curves. m,t-curves are calculated and displayed for information.

 Special display option for splitter blades:
The display of main and splitter curves can be toggled by the check boxes independently.

Depending on the selected blade shape (see Blade properties 292) the design of the mean lines is more or less restricted.

- Freeform blades, 2D blades, Radial element blades 323
- Circular blades, Straight blades 327

For some blade shapes, user defined angular positions can be loaded using the Progression dialog 46.

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.

If the impeller has **Unlinked splitter blades** (see Blade properties 292), then you can specify the behavior of the splitter in case the main blade is changing:

- Rel. position to main blade is fixed
- Abs. position of splitter blade is fixed

The **Frontal view** (switch above the diagram) represents the designed mean lines in a frontal view, including diameters d_N and d_2.

Some more blade information is displayed in tables and diagrams in order to check the design and for informational purposes:

- Additional 328 Views 328

The blade lean angle can be manipulated only indirectly:

- Blade lean angle 332

Possible warnings

Problem	Possible solutions
Blade angles βB1, βB2 and meridional/ tangential blade extension could result in a nontypical blade shape.	
Blade angles βB1, βB2 and meridional/ tangential blade extension could result in an extreme blade shape.	
The values of the blade angles βB1, βB2 and the meridional and tangential blade extension most likely result in an abnormal or strange blade shape. To avoid any subsequent problems such mean line shapes are blocked.	In theses cases the blade is highly curved or has a S-shape. To design a reasonable blade the wrap angle has to be not too low and not too high. You can a) modify the blade wrap angle φ (checking the blade overlapping) or b) modify the blade angles βB1 and βB2 (probably the main dimensions have to be adapted)
ΔβB1/2 (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.
ΔβB1/2 (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.
Overlapping of neighboring blades seems to be too small. Overlapping of neighboring blades seems to be too high.	
The overlapping of neighboring blades is too small/ too high.	Modify the blade wrap angle φ and/ or the number of blades (see Blade angles 307).

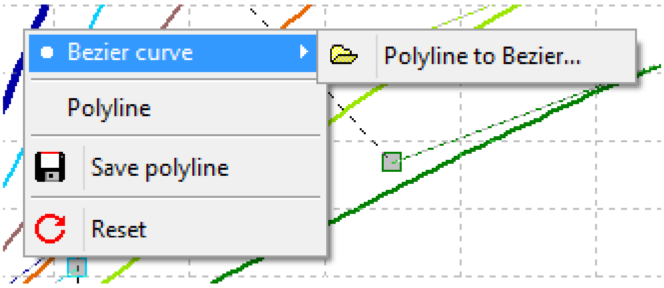
Problem	Possible solutions
High tangential leading edge sweep angle requires high number of spans.	
Leading edge sweep angle (tangential difference between hub and shroud meanline at LE) is high. This curved shape requires a minimal number of spans to avoid abnormal or strange blade shape. A warning level and an error level exist for this test.	Increase the number of spans - see Blade angles 307.
It's not possible to keep the meridional boundary conditions for this blade shape.	
r, z coordinates at leading/ trailing edge of one or more mean lines do not correspond to their meridional positions.	Check meridional contour, blade shape and mean lines.

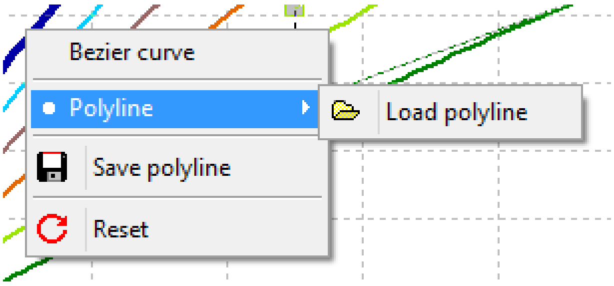
8.3.2.1 Freeform blades, 2D blades, Radial element blades

Freeform blades have the highest flexibility - the mean lines of all blade profile can be designed directly.

For 2D blades and radial element blades you can design the hub mean line only, all other mean lines are calculated automatically due to the constraints of the blade shape.

In general the mean lines are represented by 3rd order Bezier curves. Using the context menu of the mean lines Bezier curves can be fitted from polylines. Moreover, the curve mode can be switched to polyline to use a user-defined polyline directly.





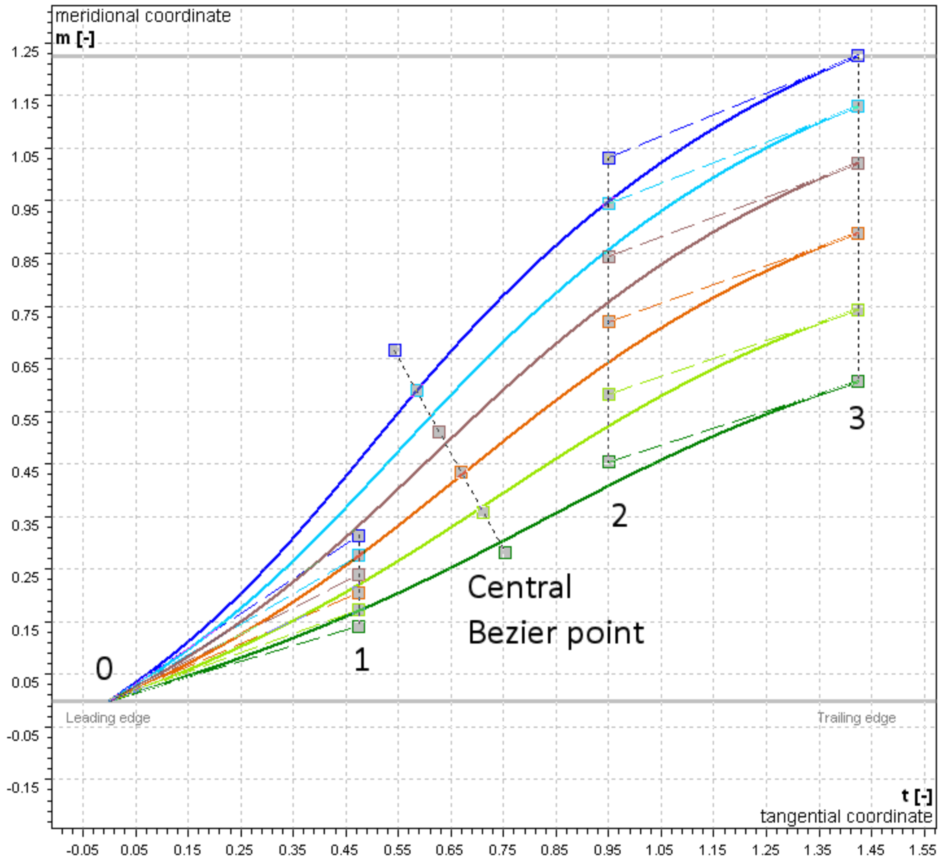
Constraints are:

- Meridional extension dm (see Meridional contour 268)
- Start angle φ_0
- Wrap angle $\Delta\varphi$

Start angle φ_0 defines the starting point of the mean lines. The absolute value is irrelevant, only the position of the mean lines to each other can be influenced. If all mean lines have the same starting point then the leading edge starts on the same angular position on all mean lines (radial leading edge). On panel **Leading edge points** you can select, if the position of points 0 of the mean lines is **Constant**, **Linear** or **User defined**.

Wrap angle $\Delta\varphi$ can be specified numerically for inner (hub) and outer (shroud) mean line, in between the values are interpolated.

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, inclusive individual wrap angles $\Delta\varphi$.



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 (dm) and the wrap angle $\Delta\varphi$. The initial wrap angle $\Delta\varphi$ is based on empirical functions 145.

In case of **Splitter blades** the design options depends on the link between main and splitter blades in the **Blade properties** 307. 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. Under **Constraints** the relative position of the splitter blade between two main blades can be adjusted. It couldn't be set on all profiles **user defined** like the Start angle φ_0.

If main and splitter blades are not linked there are all degrees of freedom in design for both.

The m-t-view of the splitter blades is shown on a separate tab (**Splitter blade (m-t)**). Additionally the profiles of the contiguous blades are shown. By default they are positioned relatively by their m-coordinate. That can be changed under **Display options** by selecting another **Splitter to main position (m-t)**.

In case of **Turbines** the situation is vice versa: The leading edge is located at high meridional coordinates whereas the trailing edge is at zero.

The wrap angle φ is initially constant for all cross sections, but it can be modified individually. The wrap angle tremendously influences the blade angle progression (β_B) along the mean line. Beta-progression can be viewed in a separate diagram.

Two points in the middle, 1 and 2, must be on a straight line at an angle of β_{B1} or β_{B2} to the horizontal in order to fulfill the boundary condition: β = dm/dt

The primary design shows points 2 at 1/4 of the wrap angle, and points 1 at 3/4.

Individual mean lines can be designed separately. 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. On the other hand, 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 β_{B1} or β_{B2} (rotation of the connecting line). Points 0 (leading edge) and 3 (trailing edge) can only be moved horizontally (m=const). Points 3 can be moved interactively (move/ rotate trailing edge). Points 0 (leading edge) can moved only by modifying wrap angles in table **Boundary conditions**.

By activating the **Central Bezier point** option a flexible central point is added for representing each mean line by a 4th order Bezier curve. As a result more flexibility is provided.

In panel **Blade angles** the blade angles β_{B1}, β_{B2} (see Blade properties 292) and the angles in x,y-plane (frontal view) $\beta_{B1,xy}$, $\beta_{B2,xy}$ are stated for information.

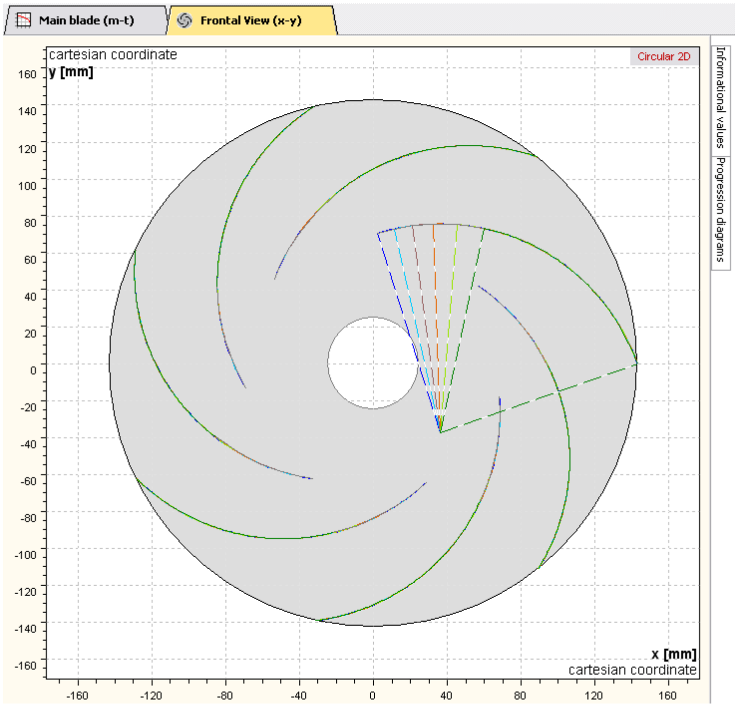
In panel **Blade information** the angles of overlap of neighboring blades φ_B and the incidence angle i (see Blade properties 292) are stated.

Possible warnings

Problem	Possible solutions
Coupling partially deactivated. Blade surface could be deformed.	
The mean lines are currently not linearly coupled, which can result in deformed blade surfaces. Either linear coupling has been deactivated or it is impossible because of highly deviating blade angle values. The warning occurs because the intersection of β_{B2} line and intersection line for one or more mean lines cannot be determined. Usually this has one of the following causes:	Activate linear coupling if it is deactivated. Homogenize β_{B2} blade angle values (see Blade properties 292).

Problem	Possible solutions
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%).	

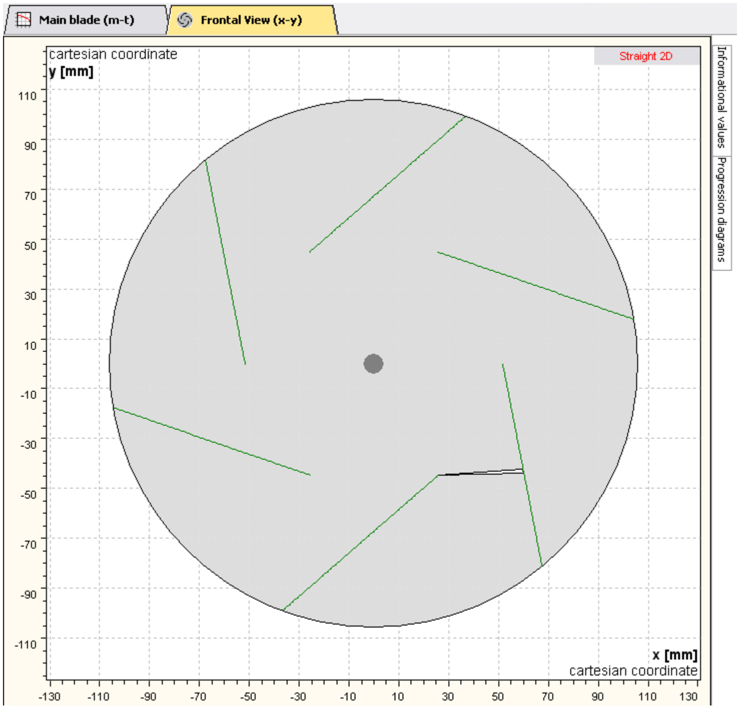
8.3.2.2 Circular blades, Straight blades



For these simple 2D blade shapes all mean lines are completely determined by blade shape and blade angles. All mean lines are computed fully automatically, so they can't be modified interactively.

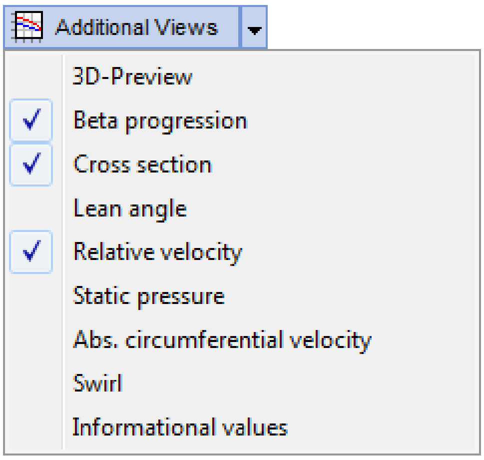
The blades are displayed in **Frontal view** most reasonable.

For circular blades the center of the circle and the blade radius are displayed in the frontal view. Furthermore the appropriate numerical values are displayed in the **Circular blade** table in the **Informational values** area (see Additional views 328).



8.3.2.3 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.

3D-Preview

3D model 172 of the currently designed mean surface.

Beta progression

β_B progression along every mean line.

Too high local extreme values should be avoided if possible.

Blade passage area

Progression of the blade passage area within a channel built by two neighboring mean surfaces as well as hub and shroud.

Lean angle

Distribution of the lean angle λ.

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 307 if just two mean lines were chosen. Otherwise the quasi-orthogonal is not displayed but internally determined by connecting corresponding points on hub and shroud mean line.

➜ see Blade lean angle 332

Relative velocity

➜ See Blade loading calculation 334

Static pressure

➜ See Blade loading calculation 334

Abs. circumferential velocity

➜ See Blade loading calculation 334

Swirl

➜ See Blade loading calculation 334

Blade loading

➜ See Blade loading calculation 334

Informational values

The tables contain additional values for information:

Radial diffuser **[Stator type "Radial diffuser" only]**

Various values to verify the quality of the diffuser design.

➜ see Mean line 394 design for "Radial diffuser" stator type

Cross section

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 (λ_1) and trailing edge (λ_2).

➜ see Blade lean angle 332

Blade loading **[Pump impeller only]**

Blade loading estimation with lift coefficient (Guelich):

Blade angle

Table with the blade angles β_B calculated in the Blade properties 292 dialog or computed due to simple blade shapes.

Blade angle in x-y

Table with the blade angles of the frontal view $\beta_{B,xy}$.

In case of strictly radial impellers these values are consistent with the blade angles β_B.

Blade angle with sine rule [Turbine rotors only]

Calculated blade angle using the sine rule.

For every mean line the calculated angles as well as their differences to the actual blade angles are given in a table.

➜ see Sine rule 331

Blade length and solidity

Table with:

- length of the blade mean lines in 3D

- solidity of the blade mean lines (chord length divided by ($\pi \cdot d_2/z$))

Other information

Table with:

- resulting angles of overlapping φ_B of 2 neighboring blades

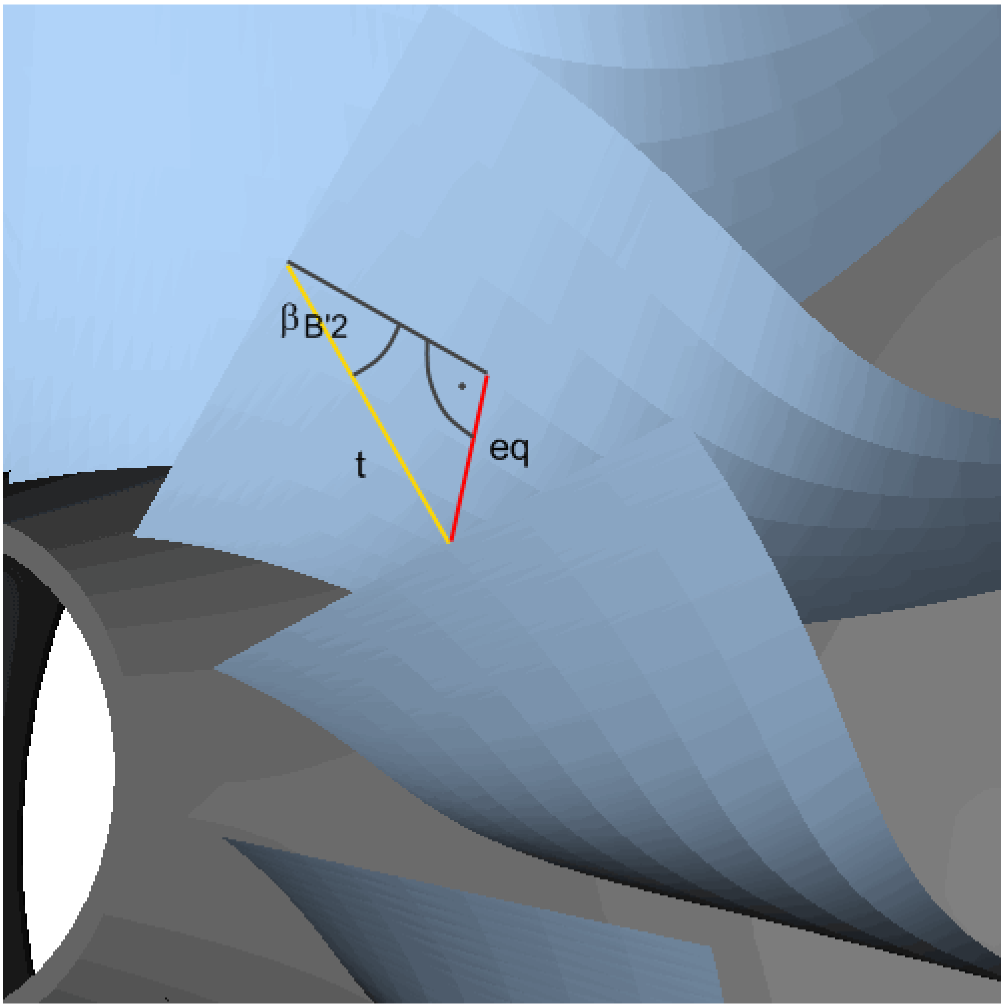
- incidence angle i for hub and shroud

8.3.2.3.1 Sine rule

[Turbine rotors only]

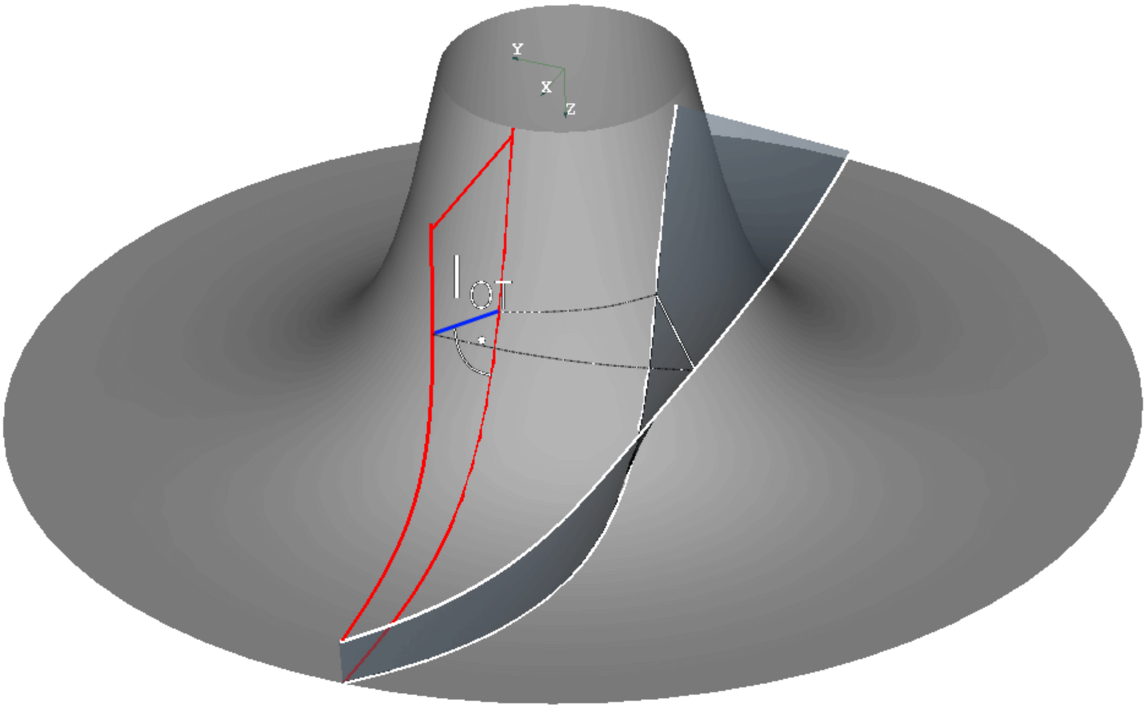
With the help of the sine rule blade angles at the outlet can be evaluated. In accordance to this rule blade angles at the outlet should have almost the same size as the angle that is built by a hypotenuse being the pitch t, and a cathetus (opposite leg) being the smallest distance between two neighboring mean lines eq at a flow surface. If this is the case the outflow can be regarded as almost tangential to the trailing edge.

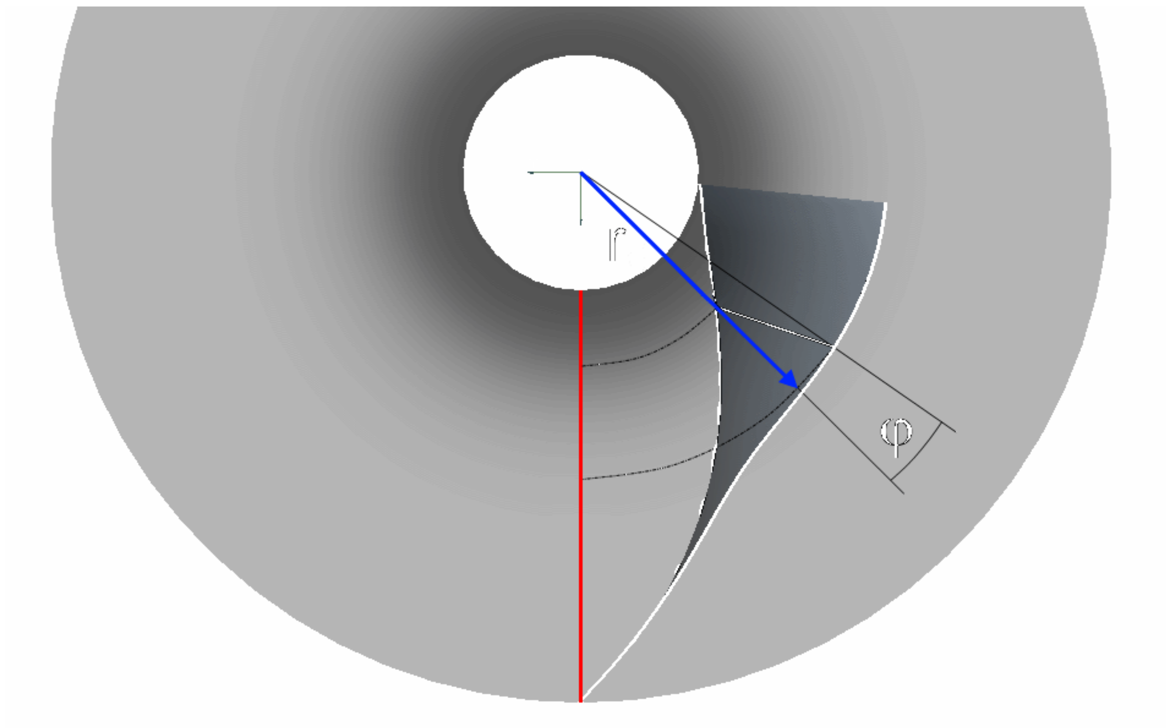
This is shown in a picture for a single mean line.



8.3.2.3.2 Blade lean angle

The blade lean angle l can not be controlled directly. It is influenced by the meridional contour, the meridional extension, the wrap angle and the mean lines. It is calculated on the basis of the length of the quasi-orthogonal l_{OT} and a radius r multiplied with the turning angle φ. The radius is that at the intersection of the quasi-orthogonal and the outer span. In the case given below this span is the shroud.

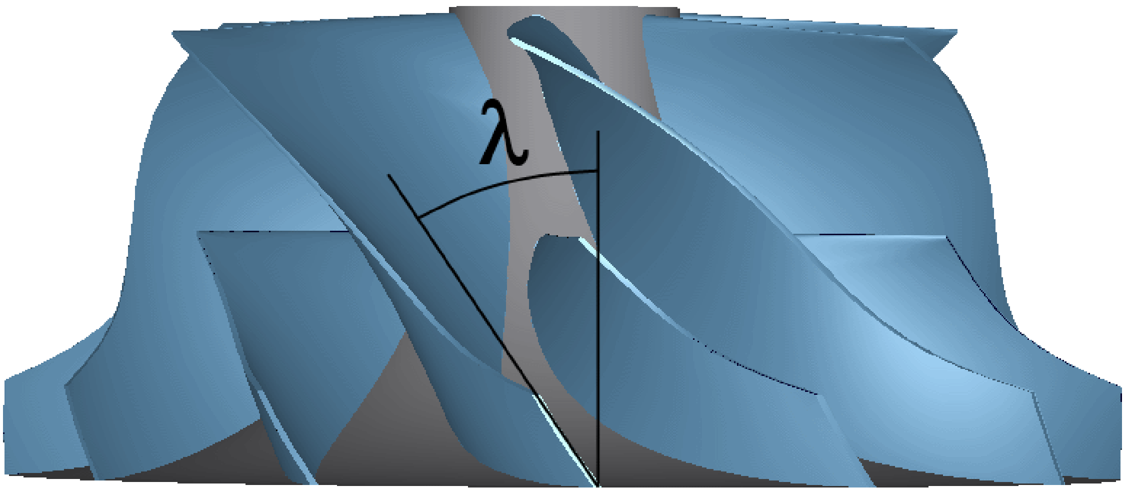




With an example of a compressor some means for the manipulation of the blade lean

angle are given:

- λ_1↑: blade angle 307 β_{B1} ↓
- λ_1↑↓: move second Bezier point 323 at leading edge
- λ_1↑: wrap angle 323 ↑
- λ_1↑ and enlargement of the curvature: reduction of the meridional extension of the meridional contour 268

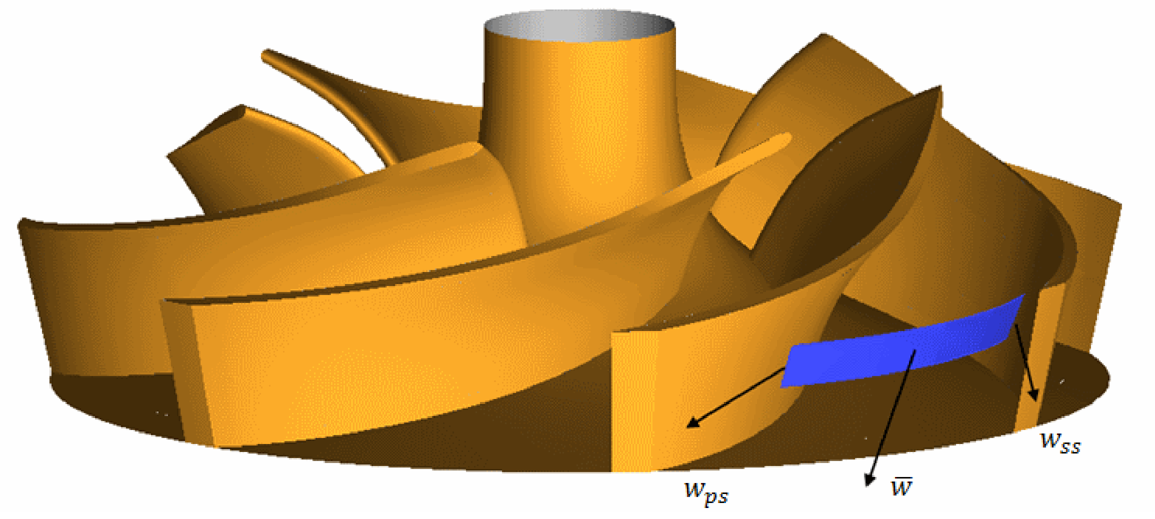


8.3.2.4 Blade loading calculation

Determination of velocity distribution on impeller blades by Stanitz & Prian 451

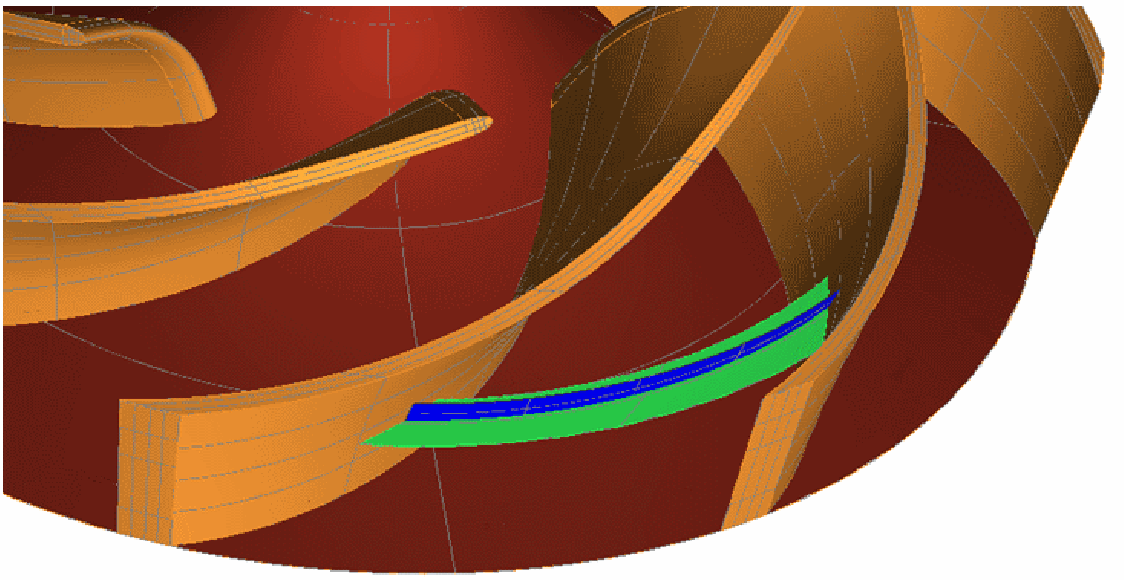
Stream lines must be known a-priori (see Meridional flow calculation 288). Stream lines rotated around z-axis build stream surfaces.The relative velocities will be calculate in a blade-to-blade section, that is encapsulated by two adjacent stream surfaces. Single values of relative velocities will be determined at r = constant. Before that an average velocity is calculated on the basis of the continuity equation:

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 Δh, the radius r, the tangential distance between pressure and suction side of two neighboring blades Δt and by a mean relative flow angle:

$$A = r \cdot \Delta t \cdot \Delta h \cdot \sin\left(\overline{\beta}\right) \cdot$$



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:

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:

$$w_{ps} = 2 \cdot \overline{w} - w_{ss} \cdot$$

Annotation

The continuity equation has to be solved iteratively for the relative velocity since the density of a compressible medium is determined by the relative velocity. The density can be calculated from isentropic relation:

$$\overline{\rho} = \rho_{t1} \cdot \left(1 - \frac{\kappa - 1}{\kappa} \frac{1}{R \cdot T_{t1}} \left(\frac{\overline{w}^2}{2} - \frac{u^2}{2}\right)\right)^{\frac{1}{\kappa - 1}} .$$

The average relative flow angle is approximated by the average value of the blade angle at suction- and pressure side. At a certain radius the assumption applies that due to the slip (decreased power) the flow cannot be considered as blade congruent anymore. The mean relative flow angle will be corrected by the slip at loci with a radius bigger than this Stanitz-Radius.

The whole procedure is based on the assumption that the flow is considered as frictionless and that shocks as well as heat transport across boundaries do not occur. There might by geometric constellations where the cross section (blue surface in the images above) is too small for the mass flow specified in the global setup 71. 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:

$$h_{ps} - h_{ss} = \frac{2\pi}{n} \cdot c_m \frac{\partial(r \cdot \overline{c}_u)}{\partial m} .$$

The blade loading can be expressed in terms of the pressure difference between suction and pressure side and divided by the total inlet pressure:

For incompressible fluids the second therm 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:

$$\frac{w_{ss} - w_{ps}}{\overline{w}}.$$

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:

$$\overline{c}_u = u - \overline{w} \cdot \cos(\overline{\beta}),$$

$$B = r \cdot \overline{c}_u.$$

Also the Ackeret 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 $0.3 \cdot w_1$:

$$Ackeret = {w_2}/{w_1},$$

$$Ackeret_{max} = 1.8 = {w_{max}}/{w_2},$$

$$Ackeret_{min} = 0.3 = {w_{min}}/{w_1}.$$

8.3.3 Blade profiles

? Impeller | Blade profiles

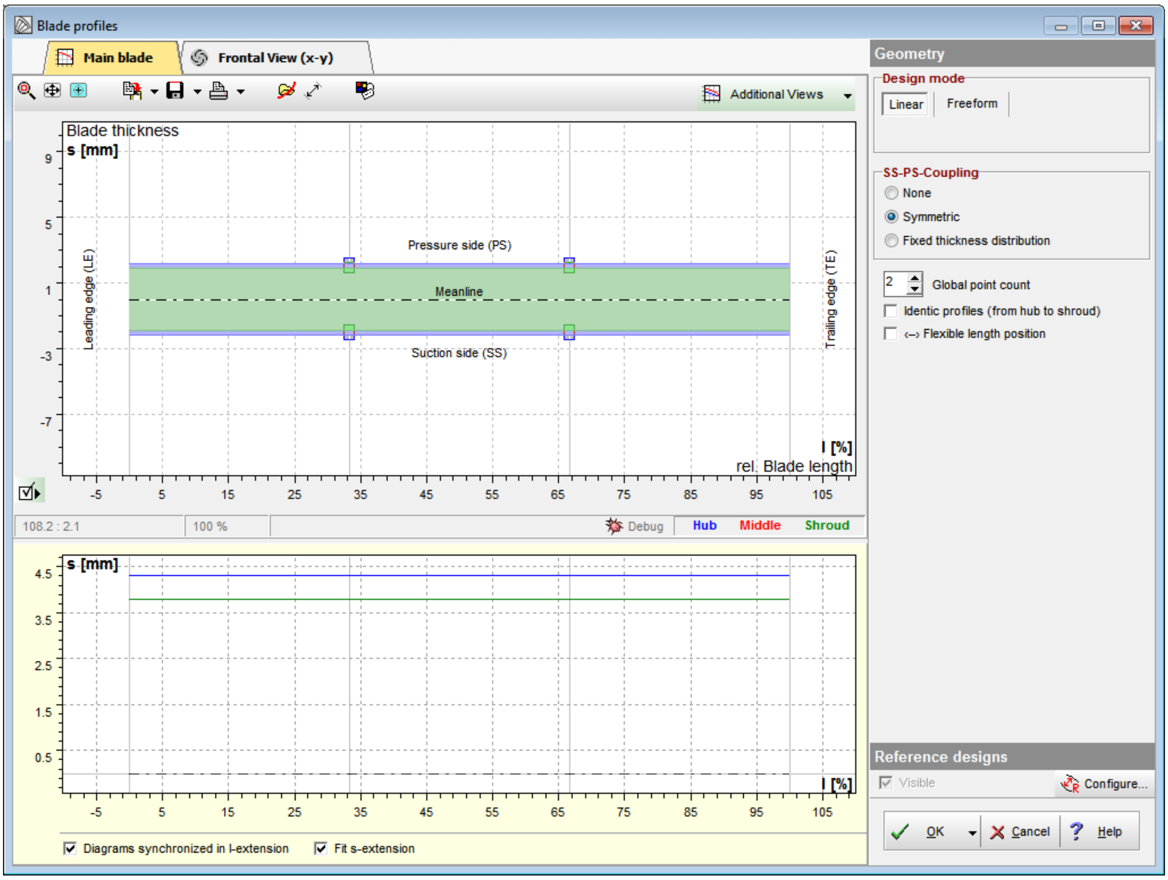
To create blade profiles (main and splitter) the orthogonal blade thickness distribution for the hub and the shroud profile is used. By default the thickness is defined at leading edge, at trailing edge and at the control points of the blade. For the initial CFturbo-design, typical values in dependence on the impeller diameter d_2 are used (see Approximation functions145).

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 orthogonal blade thickness values are added to both sides of the blade mean line to create the pressure and suction sides of the blade.

In the panel **Geometry** 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

Linked to Main
Only for splitter blades: splitter profile is linked to main profile

Global point count

Global number of control points

Identic profiles

All profiles have the same thickness distribution

Flexible length position

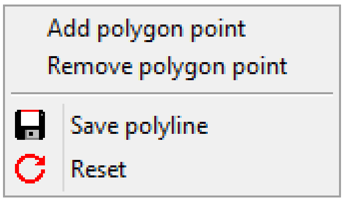
Shifting control points in horizontal direction

SS-PS-Coupling

None
No coupling between suction side and pressure side

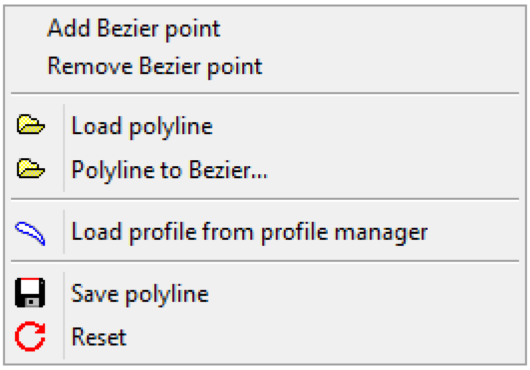
Symmetric
Symmetric thickness distribution: control points on suction and pressure side are coupled

Fixed thickness distribution
Shifting the thickness distribution to pressure/suction side whereas the distribution itself remains constant



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 343 conversion is available as well as using a thickness distribution from a pre-defined profile from profile manager 152.



Info

The Info panel represents information of the designed blade profile.

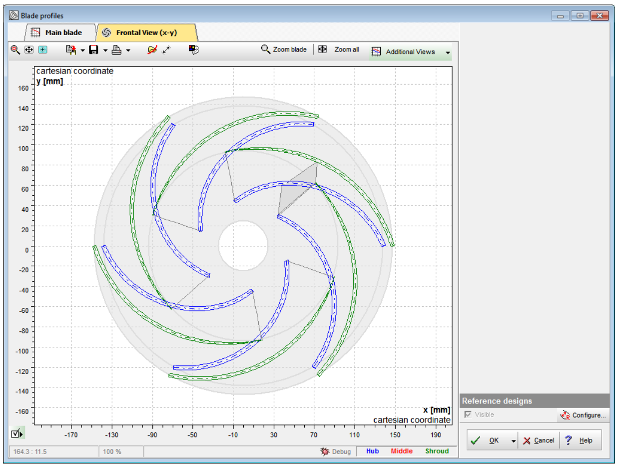
Throat area
Smallest cross section between 2 neighboring blades

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 292 dialog.
Please note that the blade thickness on leading and trailing edge should be modified in the Blade properties 292 dialog only. In this case the blade angle calculation should be updated due to the blade blockage.

Display options

The Display options only influence the graphical representation. For instance, the visibility of the smallest cross section can be toggled.



The **Frontal view** (switch above the diagram) represents the designed profiles in a frontal view, including diameters d_N and d_2.

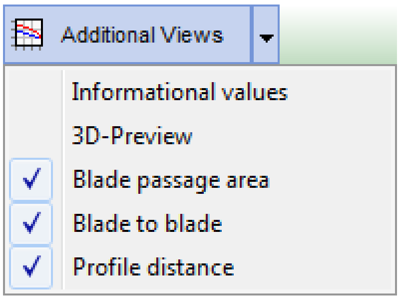
Furthermore, the smallest cross section between 2 neighboring profiles is displayed.

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
Thickness values do not match with target thickness on LE/TE	
Current profile thickness on leading- / trailing edge deviate from the specifications of the Blade properties 292 dialog.	Check the imported profile data if the values for leading and trailing edge match those of the Blade properties 292 dialog.
Internal thickness is lower than those specified for hub/shroud in blade properties.	
After changing the blade thickness on leading or trailing edge in the Blade properties 292 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

8.3.3.1 Additional views

The following information can be displayed in the blade profile dialog using the "Additional views" button:



Informational values

Some additional values are displayed for information:

- Actual thickness at hub and shroud
- Target thickness at leading and trailing edge of hub and shroud respectively

3D-Preview

3D model [172] of the currently designed blades.

The 3D-Preview contains the blades.

Blade passage area

Area that is approximately perpendicularly flown through and formed by hub, shroud and two neighboring blades.

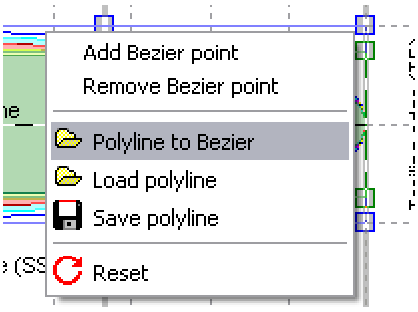
Blade to blade

Two neighboring blades in m-t-co-ordinates. In display options  the span to be displayed can be selected.

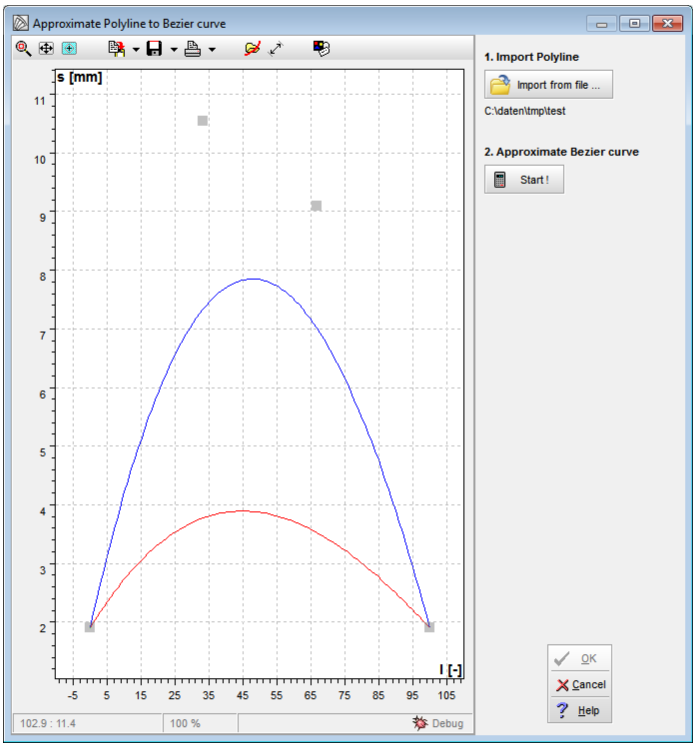
Profile distance

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.

8.3.3.2 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.

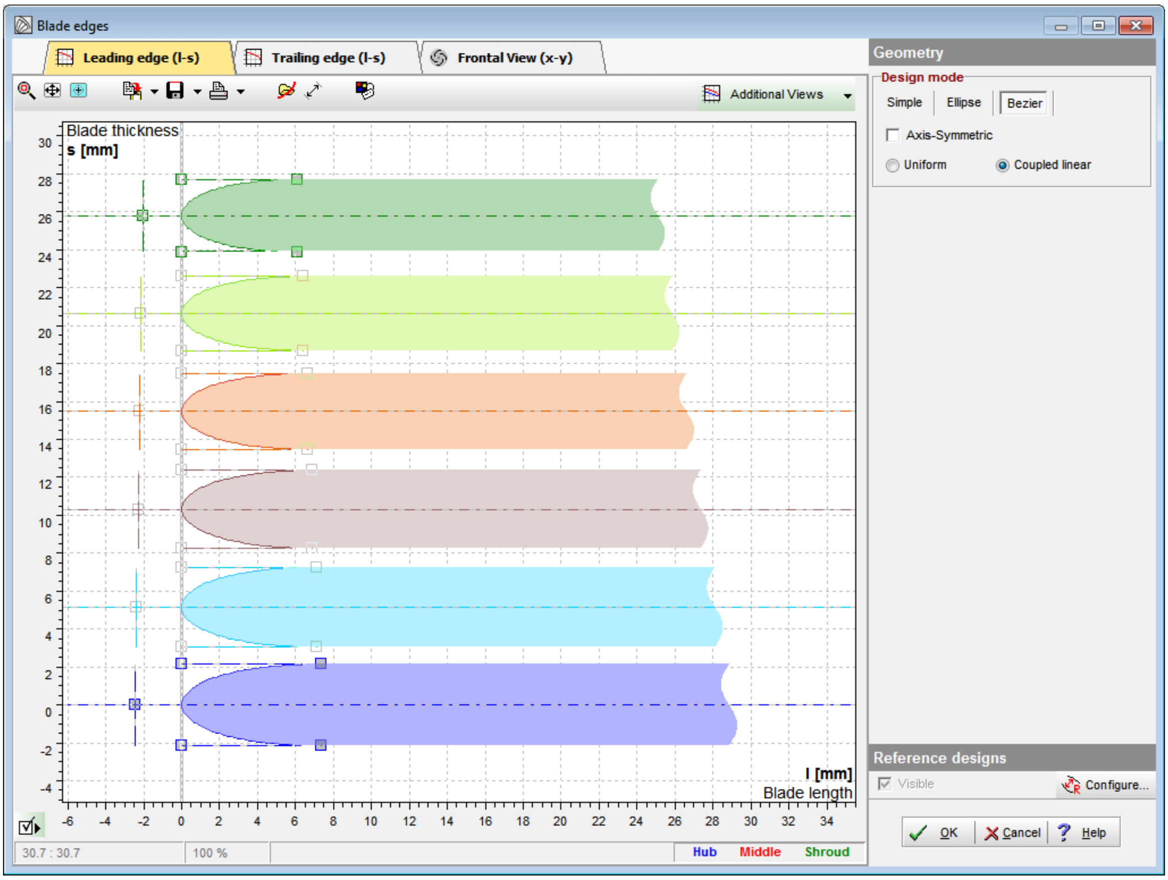
8.3.4 Blade edges

? Impeller | Blade edge

The previously designed blade has a blunt leading and trailing edge (connection line between endpoints of suction and pressure side).

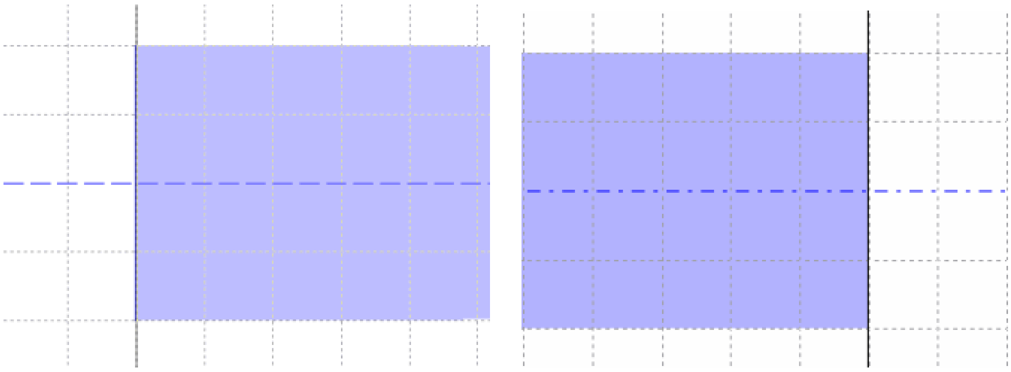
The blade edges are designed by specifying its thickness distribution. The representation of the blade thickness s is made on 15% of the straight blade length l on **leading** and **trailing edge**.

If the complete thickness distribution including leading or trailing edge was already designed in the Blade profile337 dialog, then the Edge position350 (transition from blade edge to blade suction/ pressure side) has to be defined only.



In panel **Geometry** the blade edge shape can be selected:

(1) Simple

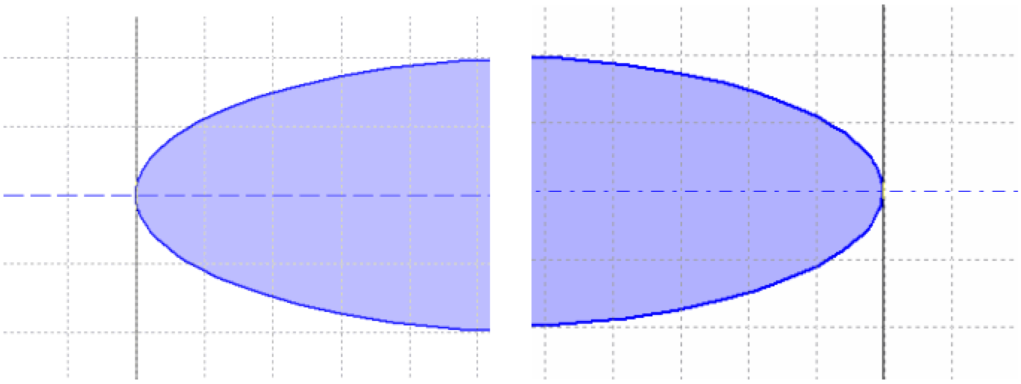


The blade edge has a blunt end.

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. The trimming is possible on the trailing edge only (or on the leading edge of turbines).

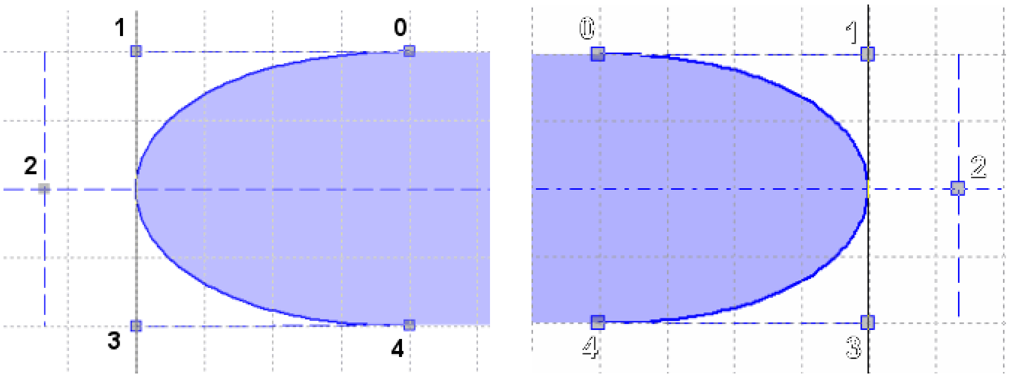
(2) Ellipse



The blade edge is rounded elliptically.

The semi**-axis ratio** can be defined. One axis runs on the mean line, the other perpendicular.

(3) 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=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

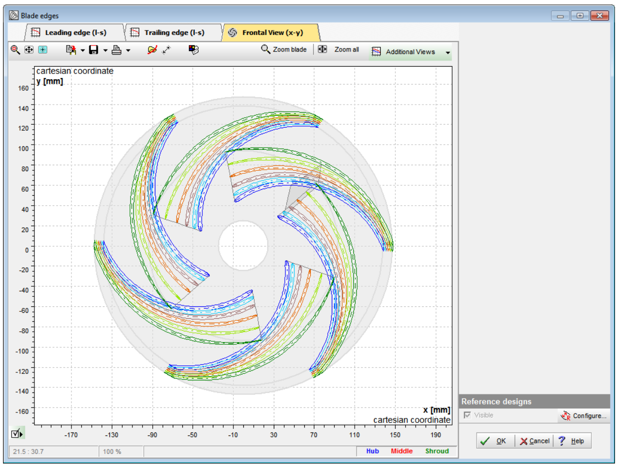
- **Uniform:**
  when designing blade edge on hub or shroud then Bezier points of all other leading edges have the same relative positions

Info

Info area represents information of **Blade edge** design.

Display options

Display options only influence the graphical representation. For instance, the visibility of the smallest cross section can be toggled.

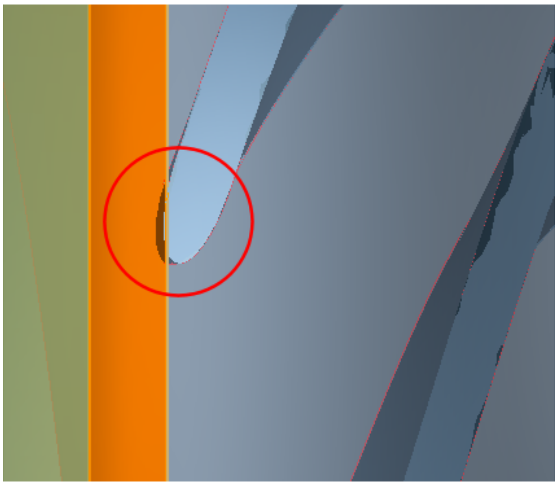


The **Frontal view** (switch above the diagram) represents the designed blades in a frontal view, including diameters d_N und d_2.

Furthermore the smallest cross section between 2 neighboring blades is displayed.

Possible warnings

<table>
<tr><th>Problem</th><th>Possible solutions</th></tr>
<tr><td colspan="2">The blade exceeds the meridional boundaries caused by the blade thickness.
Check the meridional leading/trailing edge position.
The model finishing option 'solid trimming' will not be available.</td></tr>
<tr><td>The warning indicates that some parts of the blade leading edge are outside the meridional dimensions of the component.

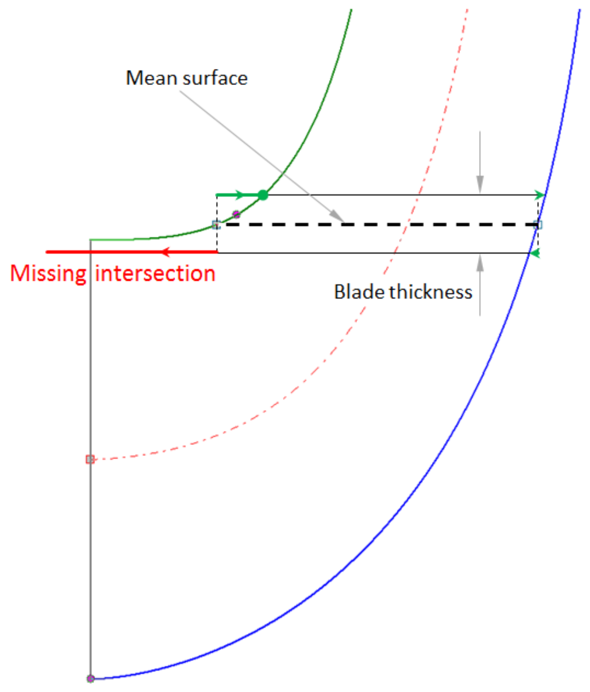
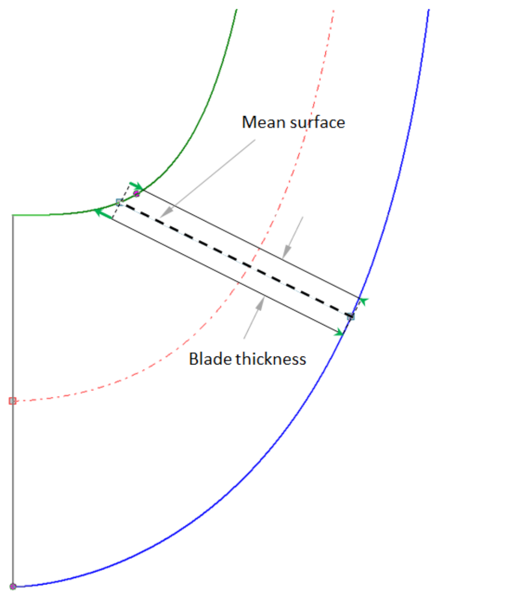
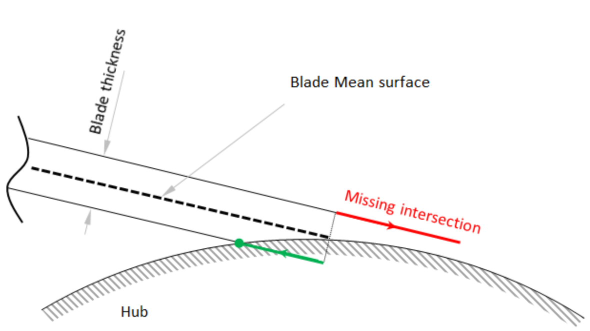
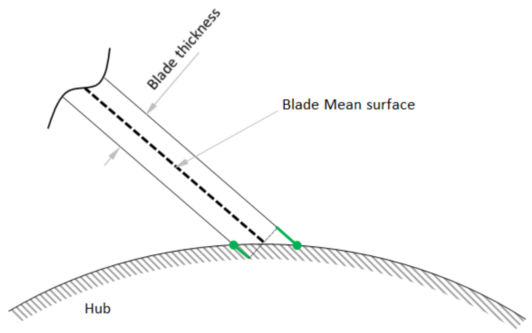


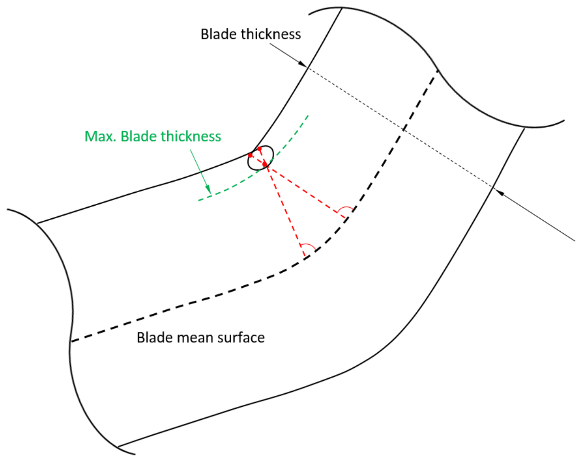
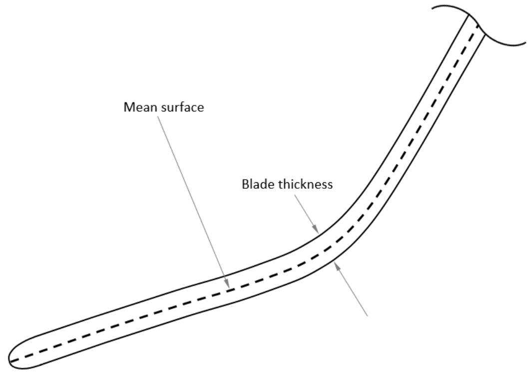
The orthogonal application of thickness on the</td><td>Dependent upon the location of these areas one has to modify leading or trailing edge.

If the leading edge (or the trailing edge of turbines) exceeds the meridional boundaries you can adjust it in the Meridional contour 268 dialog only.

Exceeding trailing edge (or leading edge of turbines) can be corrected by trim on in/ outlet.</td></tr>
</table>

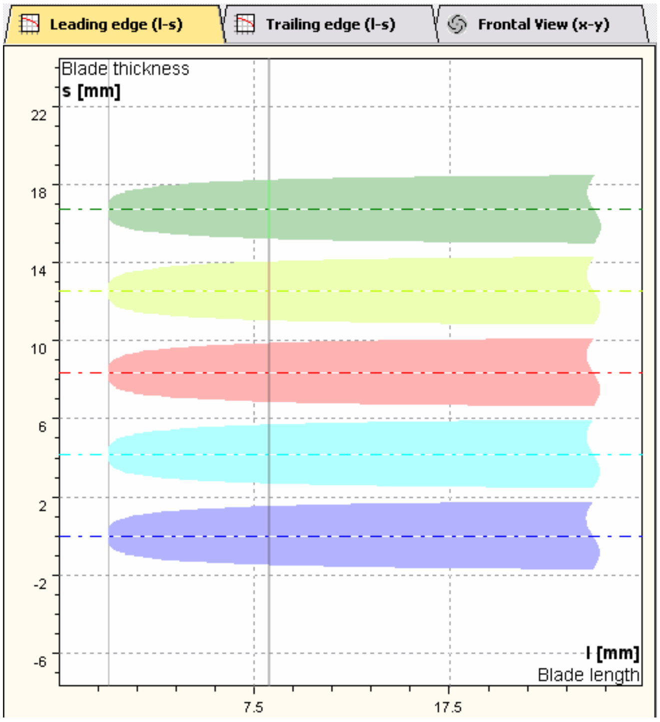
Problem	Possible solutions
mean lines can result in some blade position outside the meridional boundaries. As a result the model finishing 378 option 'solid trimming' will probably fail.	
Error while extrapolating Blade to reach Hub/ Shroud surface. Check meridional geometry, blade angles and thickness. Trim may be poor/failed, due to meridional contour at suction port and LE.	
The orthogonal blade thickness is added to the blade mean line to create the blade sides. Then one blade side will be trimmed on hub/ shroud, the other one will be extrapolated to hub/ shroud surface. For the below illustrated configurations of meridional contour and blade geometry the extrapolation fails.	Meridional contour 268: Account for blade thickness during leading edge positioning or align leading edge towards the direction of the shroud normal (see images below). The trimming/ extrapolation of blade and hub/ shroud will be successfull 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. Blade profile 337: Reduce blade thickness Mean line 319: Check mean line shape and keep lean angle on a low level

<table>
<tr><th>Problem</th><th>Possible solutions</th></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td colspan="2">Pressure/ Suction side at Hub/ Shroud:
max. thickness is too high to get smooth surface.</td></tr>
<tr><td>The combination of of high blade thickness and high meanline curvature results in degenerated blade profiles and prevents creating smooth blade surface.</td><td>Either blade thickness at the specified profile side or meanline curvature at the specified span position has to be reduced .</td></tr>
</table>

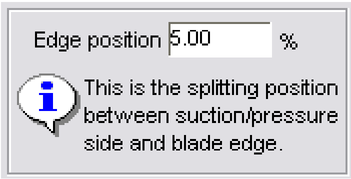
Problem	Possible solutions
	

8.3.4.1 Edge position

If the complete thickness distribution including leading or trailing edge was already designed in the Blade profile 337 dialog, then the Edge position 350 (transition from blade edge to blade suction/ pressure side) has to be defined only.



In panel **Geometry** 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%.

8.4 Airfoil/Hydrofoil design

The design of the blade's geometry is made in three steps in this design mode:

(1) Blade properties 351

(2) Blade profiles 364

(3) Blade sweeping 365

8.4.1 Blade properties

? Impeller | Blade properties

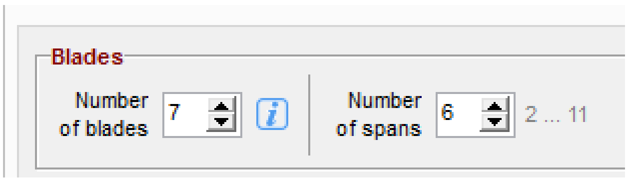
Definition of blade properties is made in three steps:

(1) Cu-specification 354

(2) Blade profile selection 357

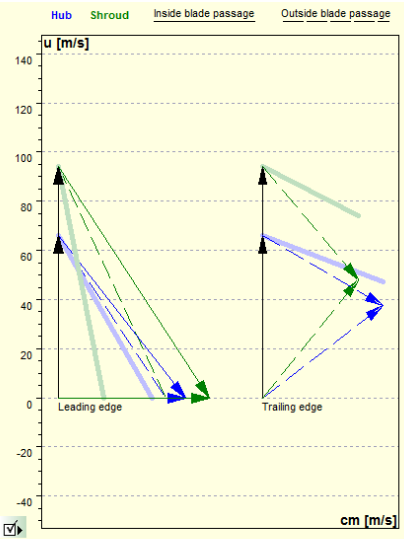
(3) Kinematics 359

Specification of number of blades and number of spans



Information

In the right panel some information are displayed which result from calculated or determined values:



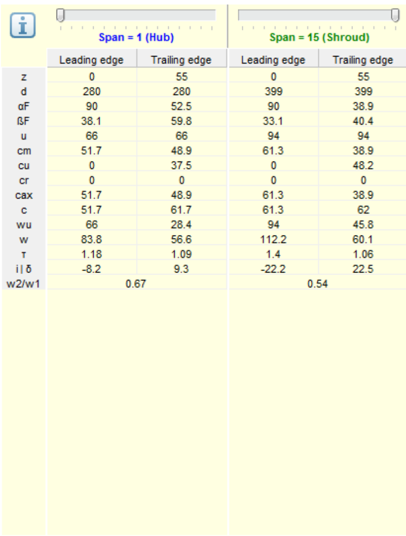
(1) Velocity triangles

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.

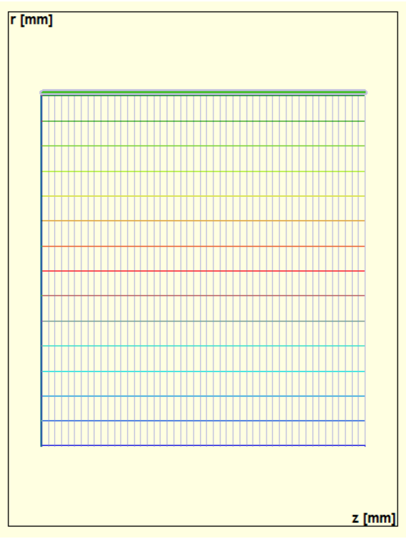


	Span = 1 (Hub)		Span = 15 (Shroud)	
	Leading edge	Trailing edge	Leading edge	Trailing edge
z	0	55	0	55
d	280	280	399	399
αF	90	52.5	90	38.9
ßF	38.1	59.8	33.1	40.4
u	66	66	94	94
cm	51.7	48.9	61.3	38.9
cu	0	37.5	0	48.2
cr	0	0	0	0
cax	51.7	48.9	61.3	38.9
c	51.7	61.7	61.3	62
wu	66	28.4	94	45.8
w	83.8	56.6	112.2	60.1
τ	1.18	1.09	1.4	1.06
i | δ	-8.2	9.3	-22.2	22.5
w2/w1	0.67		0.54	

(2) Values

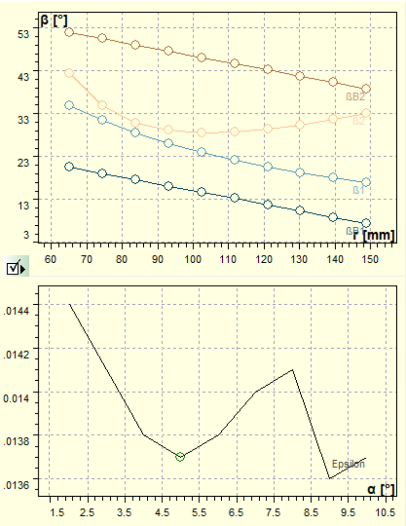
Numerical values of velocity components and flow angles are displayed in a table. The track bar on top of table can be used to get the values at any span. A short description is at mouse cursor too:

d	Diameter
α	Angle of absolute flow to circumferential direction
β	Angle of relative flow to circumferential direction
u	Circumferential velocity
c_m	Meridional velocity (c_m=w_m)
c_{ax}	Axial component of absolute velocity
c_r	Radial component of absolute velocity
c_u	Circumferential component of absolute velocity
c	Absolute velocity
w_u	Circumferential component of relative velocity: w_u+c_u=
w	Relative velocity
τ	Obstruction by blades (see below)
i	Incidence angle: $i = \beta_{B1} - \beta_1$
δ	Deviation angle: $\delta = \beta_{B2} - \beta_2$
w_R	Deceleration ratio of relative velocity: $w_R=w_2/w_1$



(3) Meridian

The Meridian with locations of the spans is displayed in this diagram.

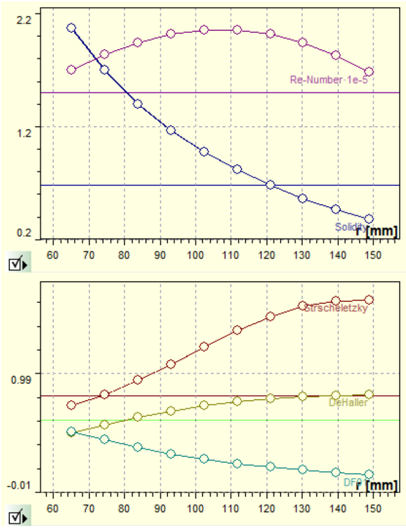


(4) Current ß

Here blade angles as well as relative flow angles are displayed versus span. Also the chosen polar together with the angle of attack is given in an additional diagram.

Progressions of geometric parameters (angles):

β1/2	Angle of relative flow to circumferential direction
βB1/2	Blade angles at leading and trailing edge



(5) Criteria

Progressions of aerodynamic and airfoil parameters:

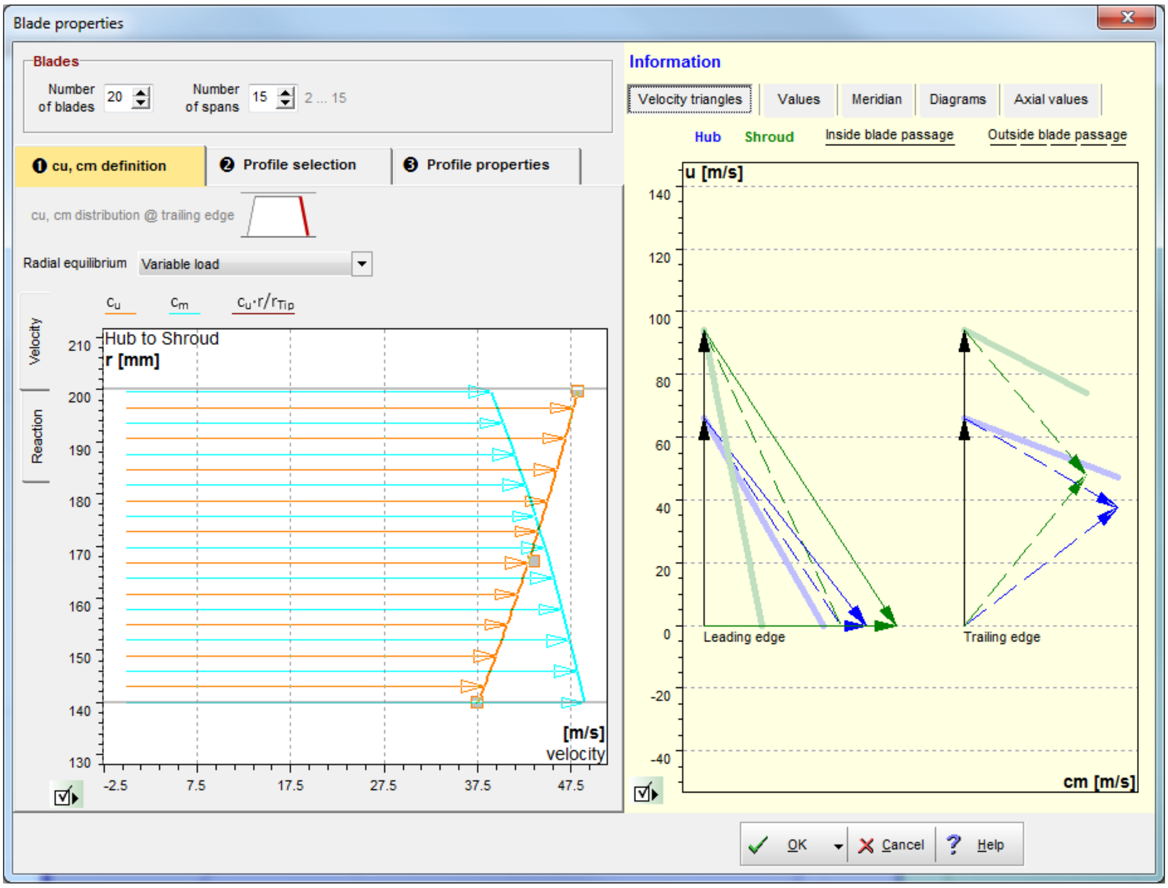
Re	Reynolds-number
l/t	solidity
DH	DeHaller critierion
ST	Strscheletzky critierion
DF01	diffusion number

8.4.1.1 Cu-specification

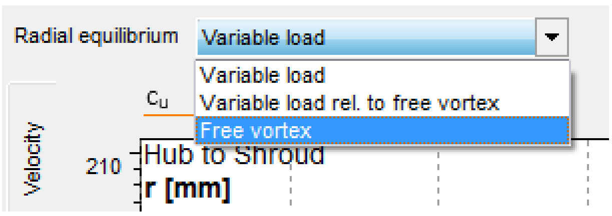
? Impeller | Blade properties

[Axial machines only]

On tabsheet **cu, cm definition** the velocity triangles at every span can be defined in accordance to the radial equilibrium 356.



It can be chosen from 3 different modes concerning the manipulation of $c_{u2}(r)$:

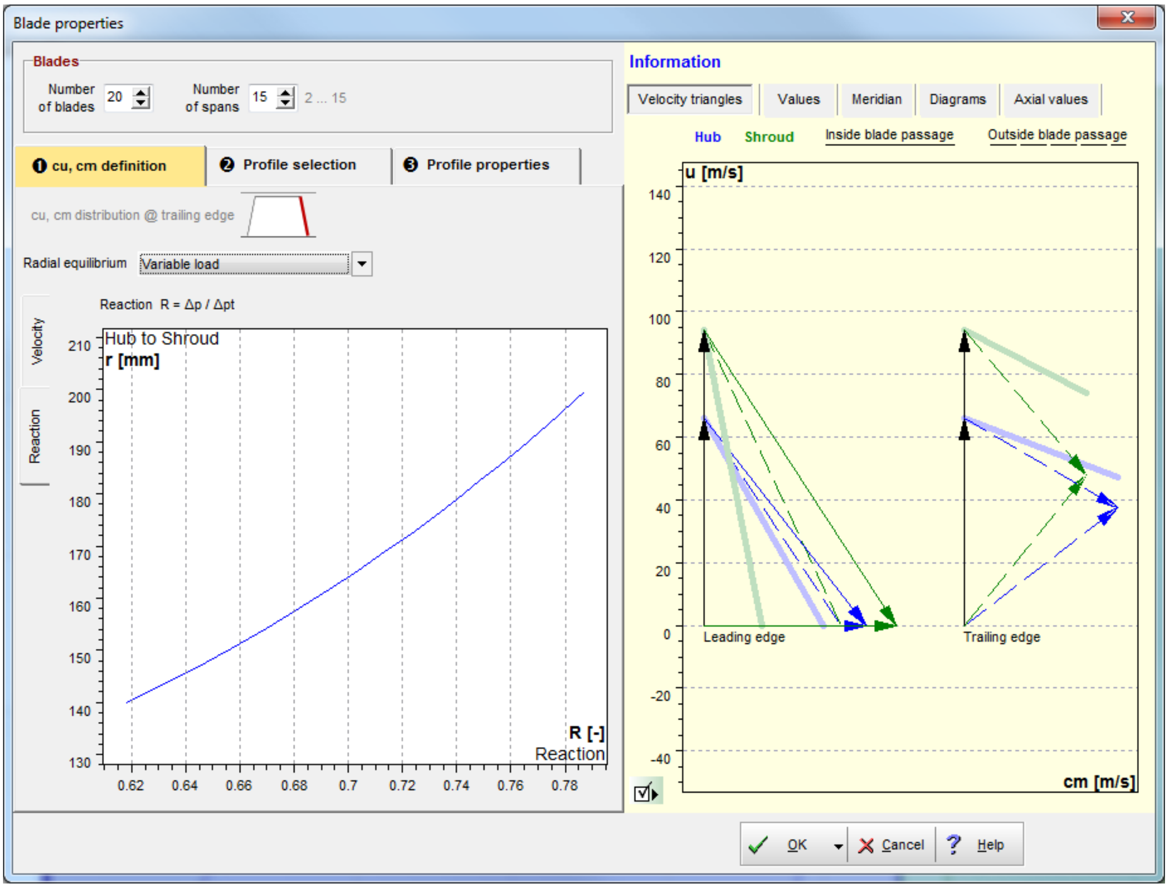


Variable load	**Free vortex**	**Variable load rel. to free vortex**
The $c_{u2}(r)$-specification is controlled by a second order Bezier curve.	$c_{u2}(r)$ is defined to get the same swirl at every span:	The slope is the derivative according to:

		$$\text{slope} = \frac{d\left(c_{u2}/c_{u2iso}\right)}{d\left(r/r_{Tip}\right)}$$ With a slope of zero a free vortex distribution is set.

Please note: There is not always a solution of the differential equation of the radial equilibrium. Therefore some Bezier point constellations are not possible.

At the second tab of the diagram the distribution of the corresponding degree of reaction is displayed: $R = \Delta h_{stat}/\Delta h_{tot}$



8.4.1.1.1 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:

$$0 = p_2 dA - (p_2 + dp_2)dA + r\varpi^2 \rho \cdot dA \cdot dr$$
$$\Rightarrow \frac{dp_2}{dr} = \rho \frac{c_{u2}^2}{r}$$

The definition of total pressure in section 2 differentiated with respect to r plus above equation yield:

$$\frac{dp_{t2}}{dr} = \rho \frac{c_{u2}^2}{r} + \rho \left(c_{m2} \frac{dc_{m2}}{dr} + c_{u2} \frac{c_{u2}^2}{dr} \right)$$

With the blade work according to Euler the equation becomes:

$$\eta_{Imp} \cdot 2\pi n \frac{d(rc_{u2})}{dr} = \frac{c_{u2}}{r} \frac{d(rc_{u2})}{dr} + c_{m2} \frac{dc_{m2}}{dr}$$

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.

$$\dot{m} = \int_{r_{Hub}}^{r_{Shr}} \rho \cdot c_{m2}(r) \cdot 2\pi r \cdot dr$$
$$P = \int_{r_{Hub}}^{r_{Shr}} u(r) \cdot c_{u2}(r) \cdot \rho \cdot c_{m2}(r) \cdot 2\pi r \cdot dr$$

From the velocity triangles the degree of reaction can be determined by the following equation:

$$R = \frac{\Delta h}{\Delta h_t} = 1 - \frac{\Delta(c^2)}{2\Delta(u_2 \cdot c_{u2})}$$

8.4.1.2 Blade profiles

? Impeller | Blade properties

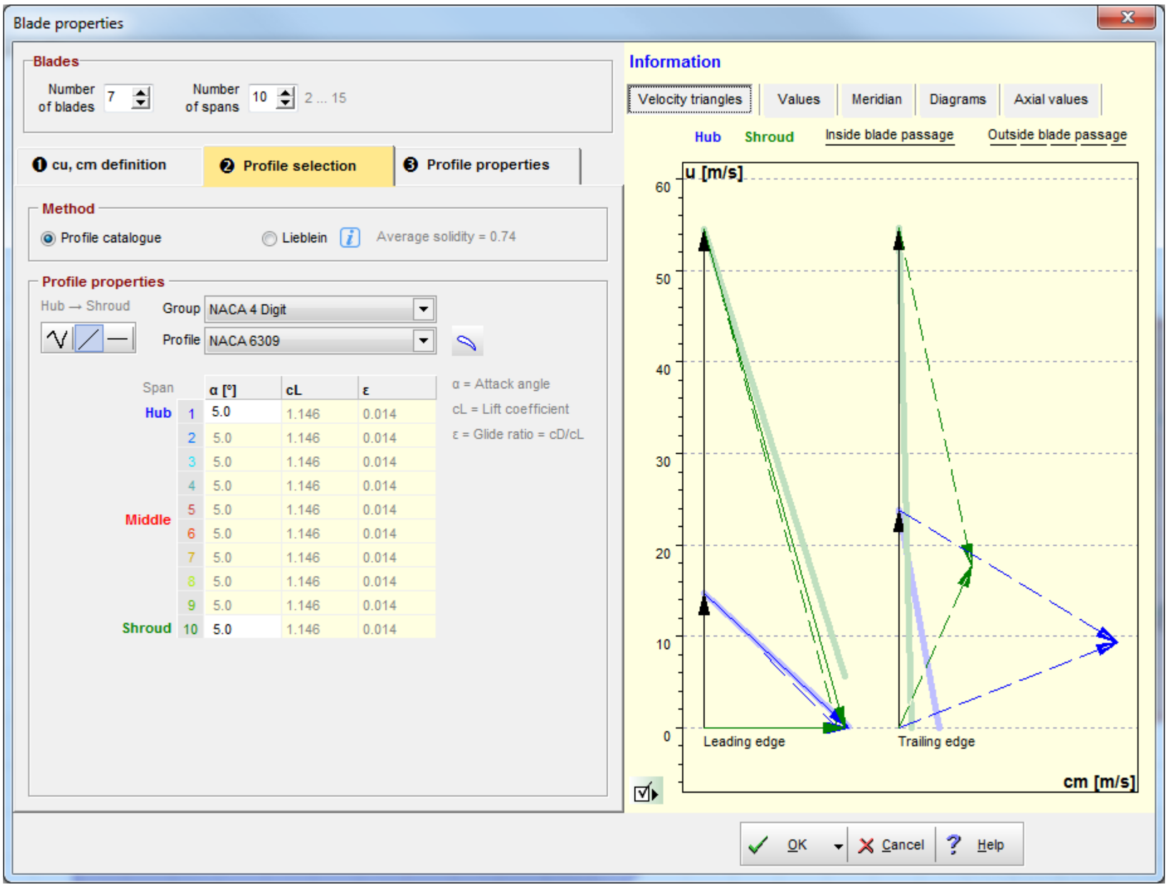
[Axial machines only]

On tabsheet **Profile selection** the axial blade profile properties are specified. To this end the profiles have to be selected from the Profile manager 152.

Two alternative methods are available:

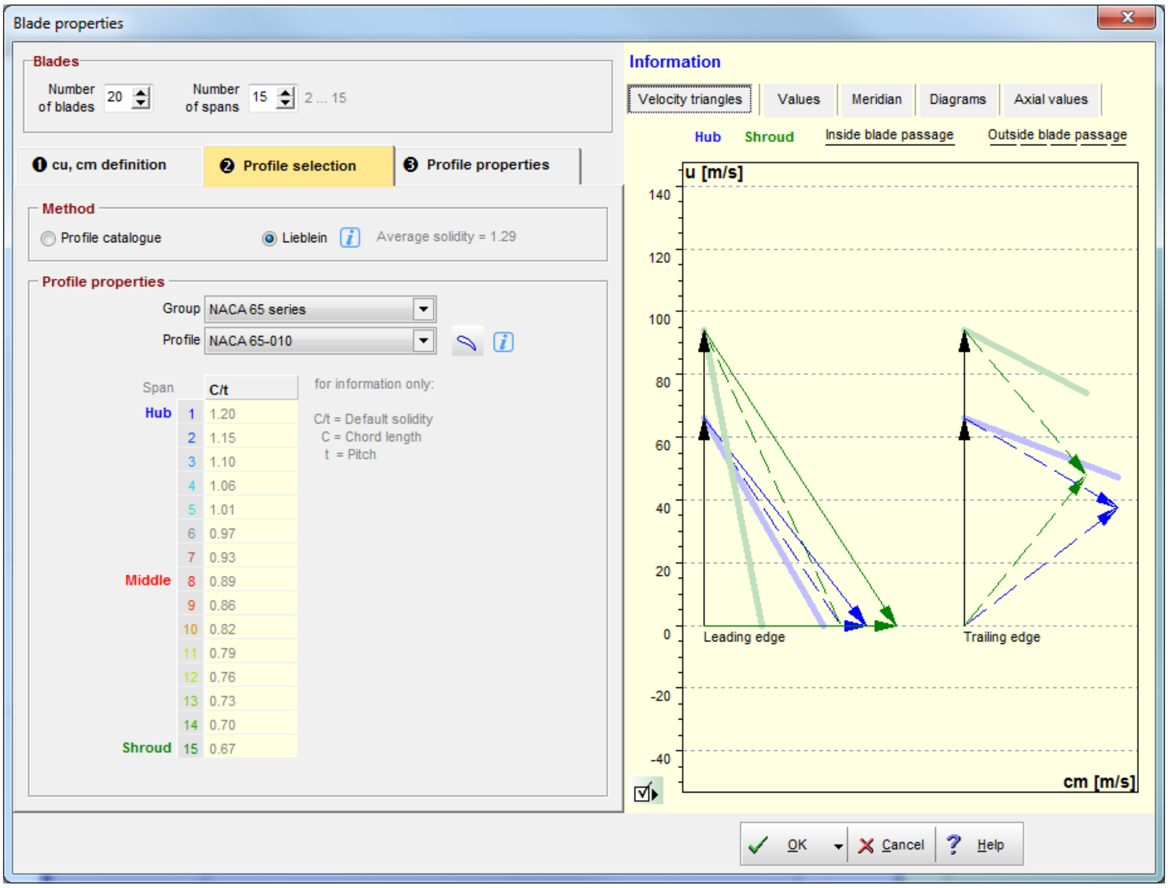
Blade element momentum method 361

Here either NACA 4 digit or point based profiles can be used. Also an angle of attack α has to be specified, see blade element momentum method 361.



Lieblein method 362

Here only profiles of the NACA 65 series can be used. A solidity has to be specified that has to be on all spans: 0.4 <= l/t <= 2.0. It is used for the calculation of the skeleton length and stagger angle, see Lieblein method 362.



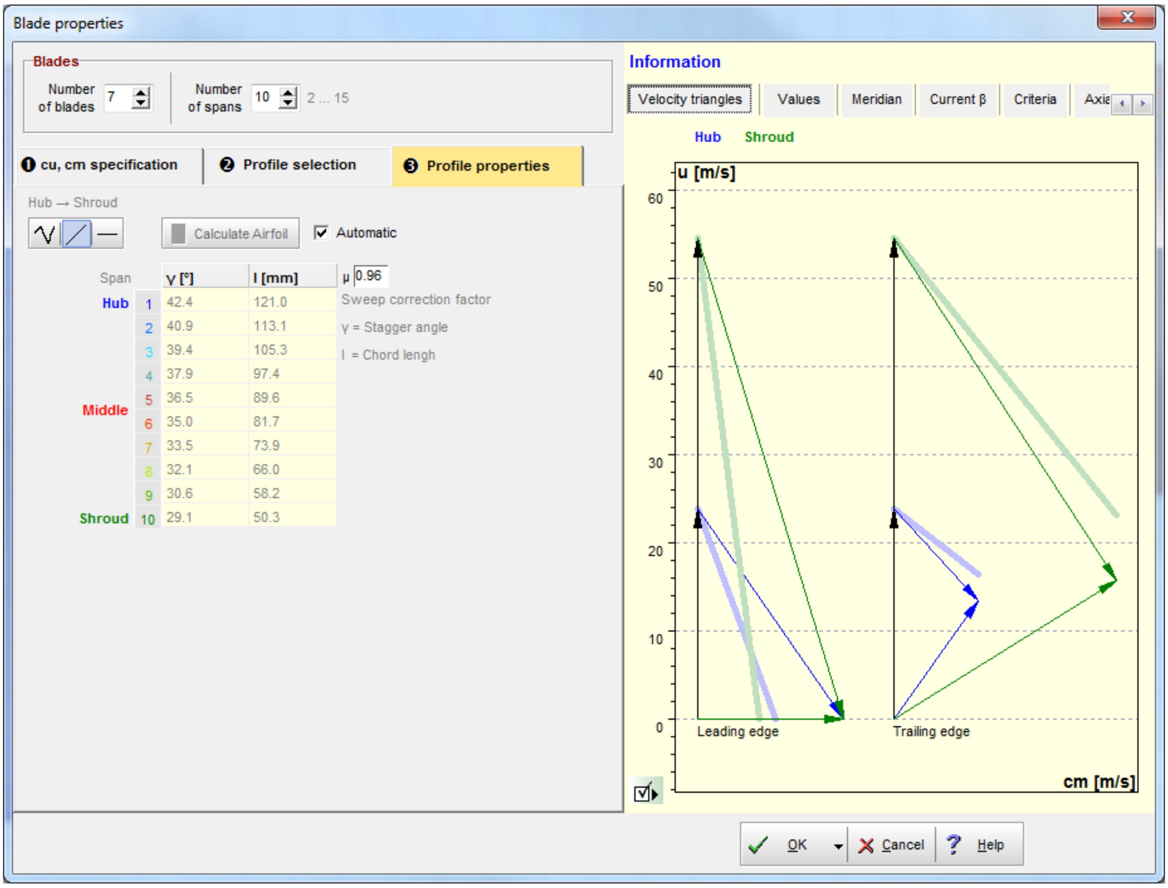
8.4.1.3 Kinematics

Two methods are available for the determination of the scaling (solidity) and staggering of the profiles:

- **Blade element momentum method** 361 **[only ventilators]**
  for low pressure applications (high specific speed nq)

- **Lieblein method** 362 **[pumps, ventilators]**
  for high pressure applications (low specific speed nq)

On the tabsheet **Profile properties** the **stagger angles** and **solidity** are calculated.



Limitations

The design methods are valid only within certain scopes:

The deceleration should no be smaller than the DeHaller 451 criterion:

$$\left.\frac{w_2}{w_1}\right|_{hub} \geq 0.6..0.75$$

.

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:

,

and for multi stage machines:

.

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 $DF_{0.1} \leq 0.6$.

8.4.1.3.1 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 s/l 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:

$$\begin{aligned} F_u &= \sin(\beta_\infty + \delta) \cdot F \\ &\approx \sin(\beta_\infty + \delta) \cdot c_L \cdot \rho \cdot \frac{w_\infty^2}{2} \cdot l \cdot b \quad \text{with} \quad F \approx F_L \end{aligned},$$

whereas if it is derived from the force balance it reads as:

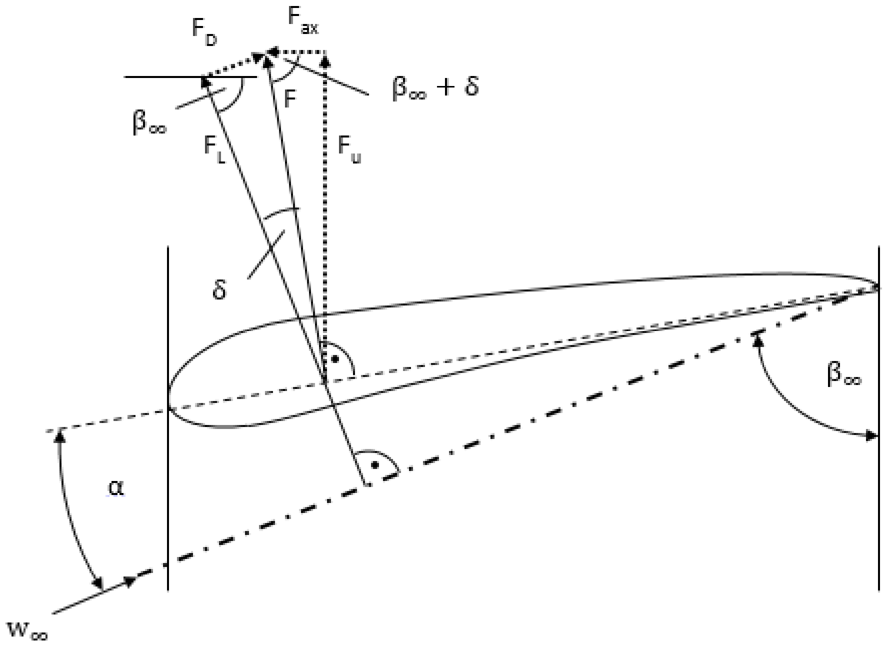
$$\begin{aligned} F_u &= \dot{m} \cdot (c_{u2} - c_{u2}) \\ &= \rho \cdot c_m \cdot t \cdot b \cdot \frac{Y_{Imp}}{u} \end{aligned}.$$

By equalizing both force descriptions one gets the following equation, which co-relates the profile properties lift coefficient c_L and solidity l/t with the design point data (Y, n, m):

$$\sin(\beta_\infty + \delta) \cdot c_L \cdot \frac{w_\infty^2}{2} \cdot \frac{l}{t} = c_m \cdot \frac{Y_{Imp}}{u}.$$

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
α	angle of attack
δ	angle between resulting force and lift force



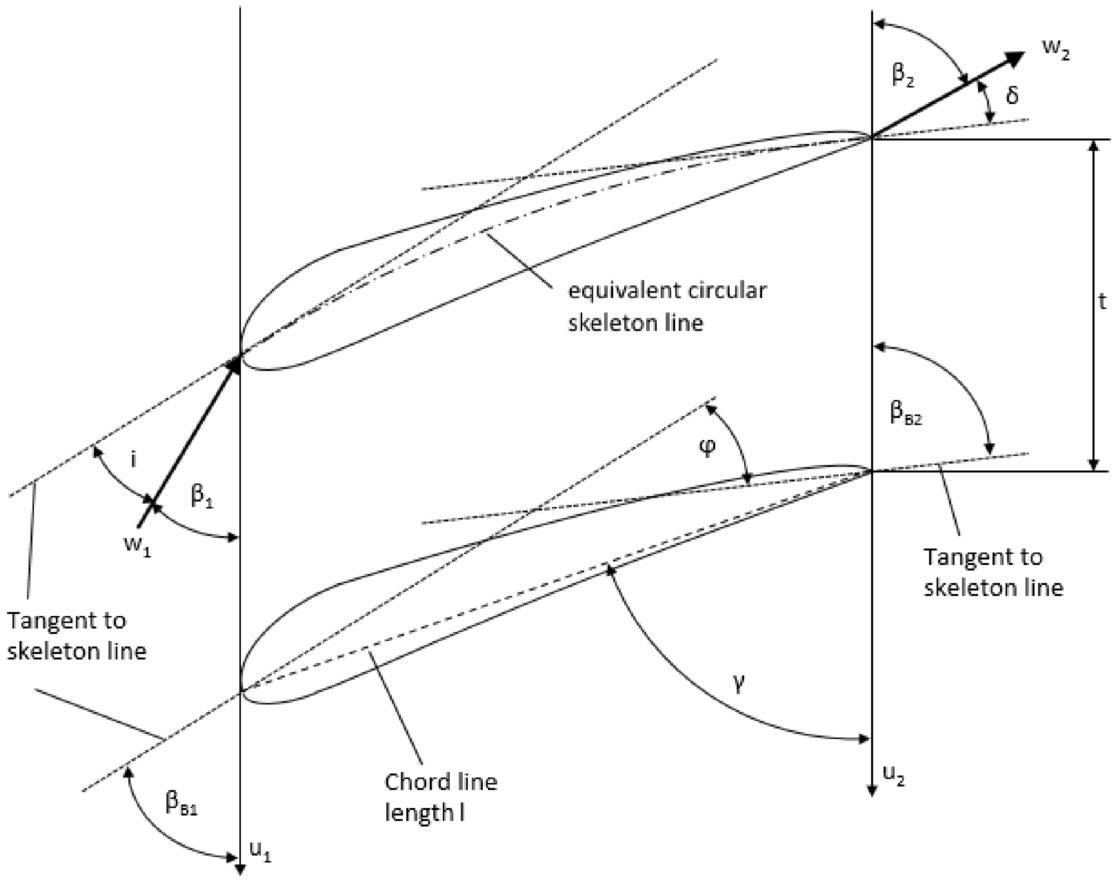
8.4.1.3.2 Lieblein method

Lieblein 451 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

γ	stagger angle
l/t	solidity (chord length/pitch)
β	Angle of relative flow
β_B	Blade angle
u	circumferential velocity
w	Relative velocity
i	Incidence angle: $i = \beta_{1B} - \beta_1$
δ	Deviation angle: $\delta = \beta_{2B} - \beta_2$

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: $0.4 <= l/t <= 2.0$



Lieblein derived design diagrams for the following parameter

- Incidence i
- Deviation δ

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 354) and the solidity l/t the incidence is determined using Lieblein's design diagrams. The same is done with respect to the deviation. Now the the blade angles at leading and trailing edge are known. Note: The blade angles are applied to the equivalent circular skeleton line with the radius:

$$r_{eq} = \frac{l}{2 \cdot \sin \frac{\beta_{B2} - \beta_{B1}}{2}}.$$

From the blade angles the stagger angle can be determined by:

.

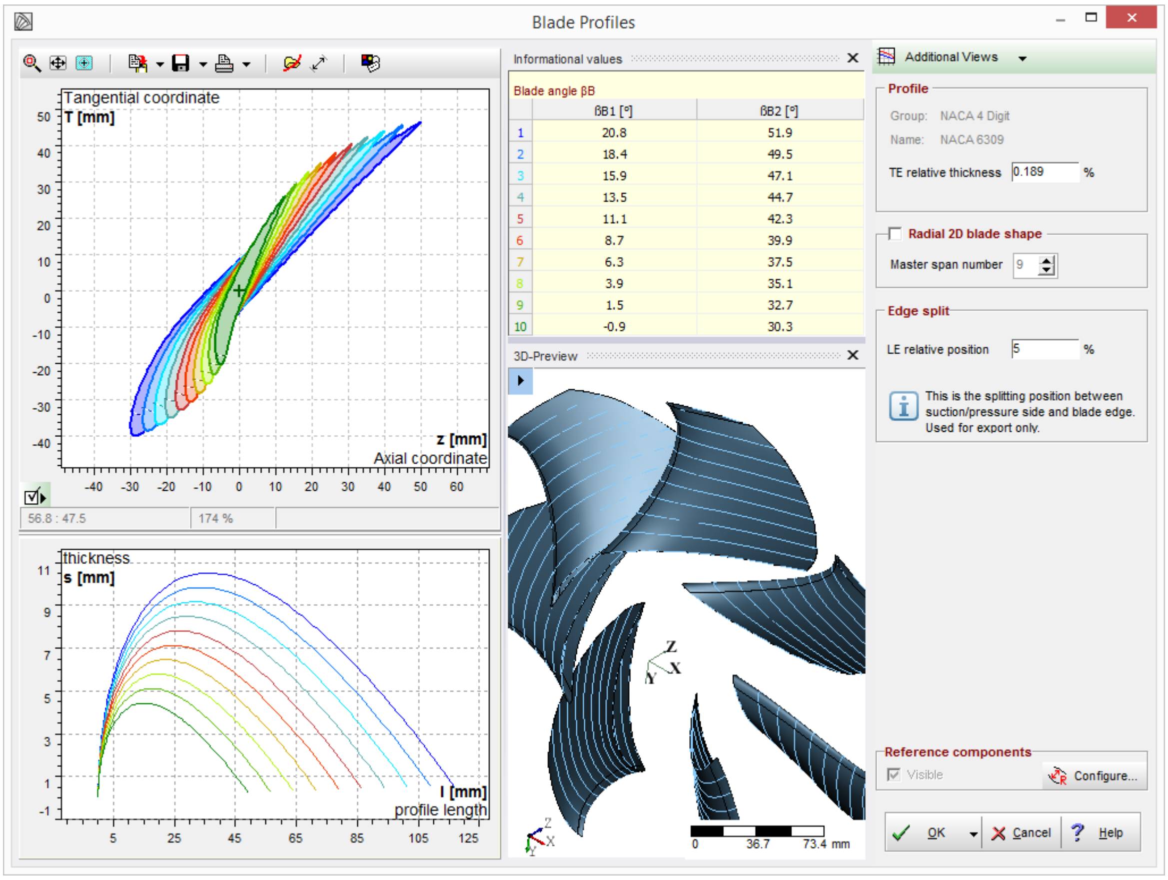
8.4.2 Blade profile

? Impeller | Blade profiles

To create 3D blade profiles the specified or calculated values from the Blade properties 351 are used:

- Profile shape based on profile selection 357
- Chord length (scaling) and Stagger angle (rotation) of each profile at the respective span position based on profile properties 359

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})
- **3D-Preview**: 3D blade shape after the 2D blade profiles were projected into its span surface

Profile

The previously selected blade profile names are displayed for information. For NACA profiles the trailing edge thickness can be adapted for manufacturing reasons. The additional thickness is added linearly over the length of the profile.

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 351.

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.

8.4.3 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.

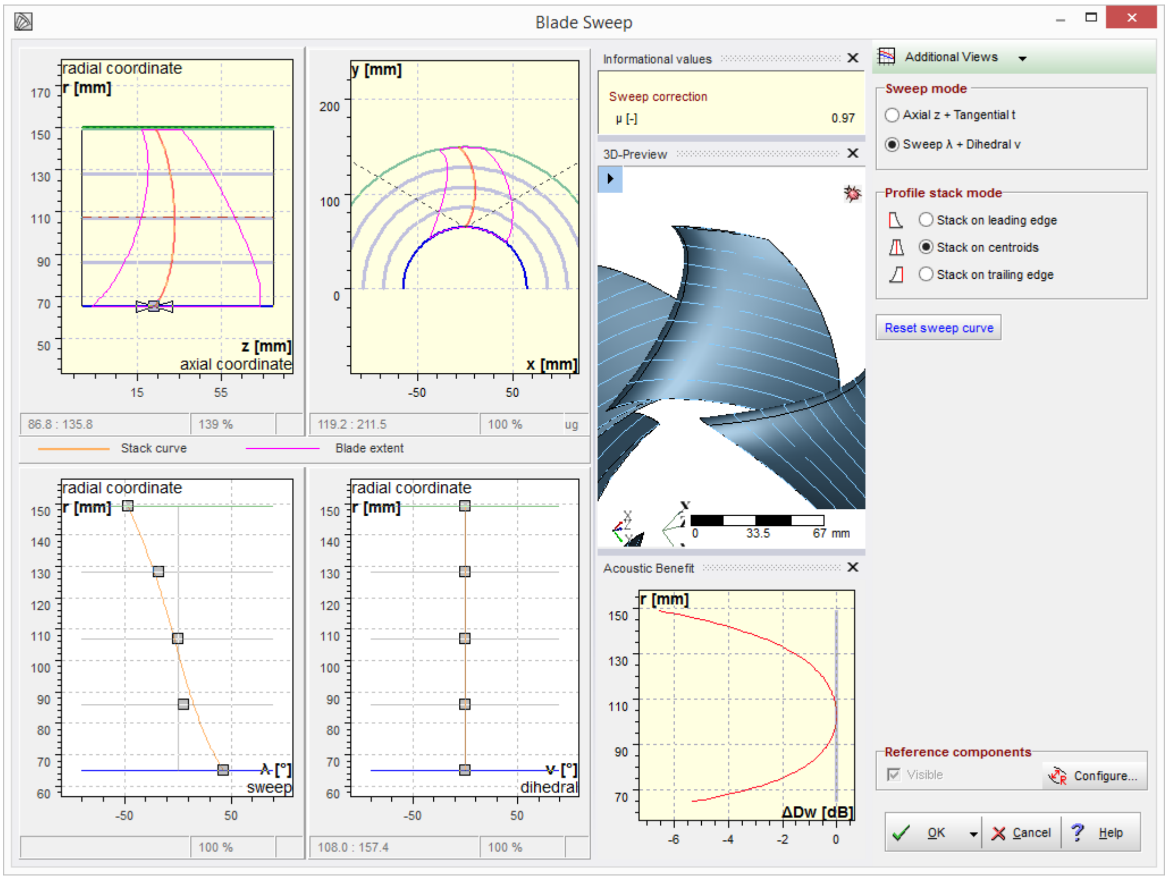
In default configuration this design step does not generate any sweep by aligning the centroid points of all profiles exactly in radial direction. You can return to an unswept configuration at any time by using the **Reset sweep curve** option.

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:

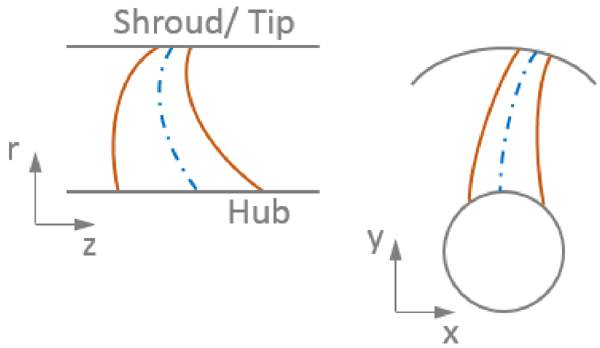
- **Informational values**: The sweep correction factor μ representing the efficiency loss by sweeping (see Kinematics 359)
- **3D-Preview**: The final result of the sweep design process, the swept 3D blade shape.
- **Acoustic benefit**: ...compared to the unswept blade design

Sweep mode

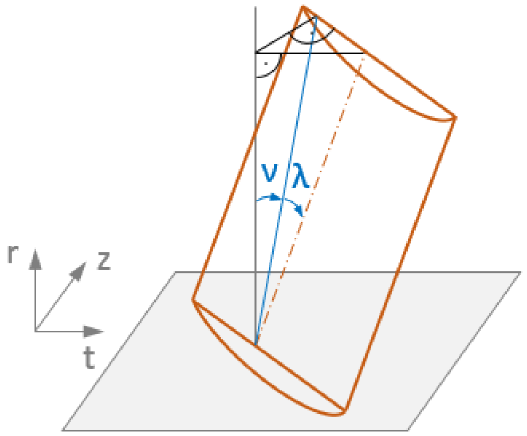
The Sweep mode controls which of the 2D Sweep projections define the blade sweep and are modifiable by the user.

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 λ + Dihedral ν (default)**
  λ: Incidence not perpendicular to blade axis, blade area nevertheless in flow direction
  ν: 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 364 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

8.5 CFD Setup

? Impeller | CFD setup

The designed geometry can be extended by **virtual** elements.

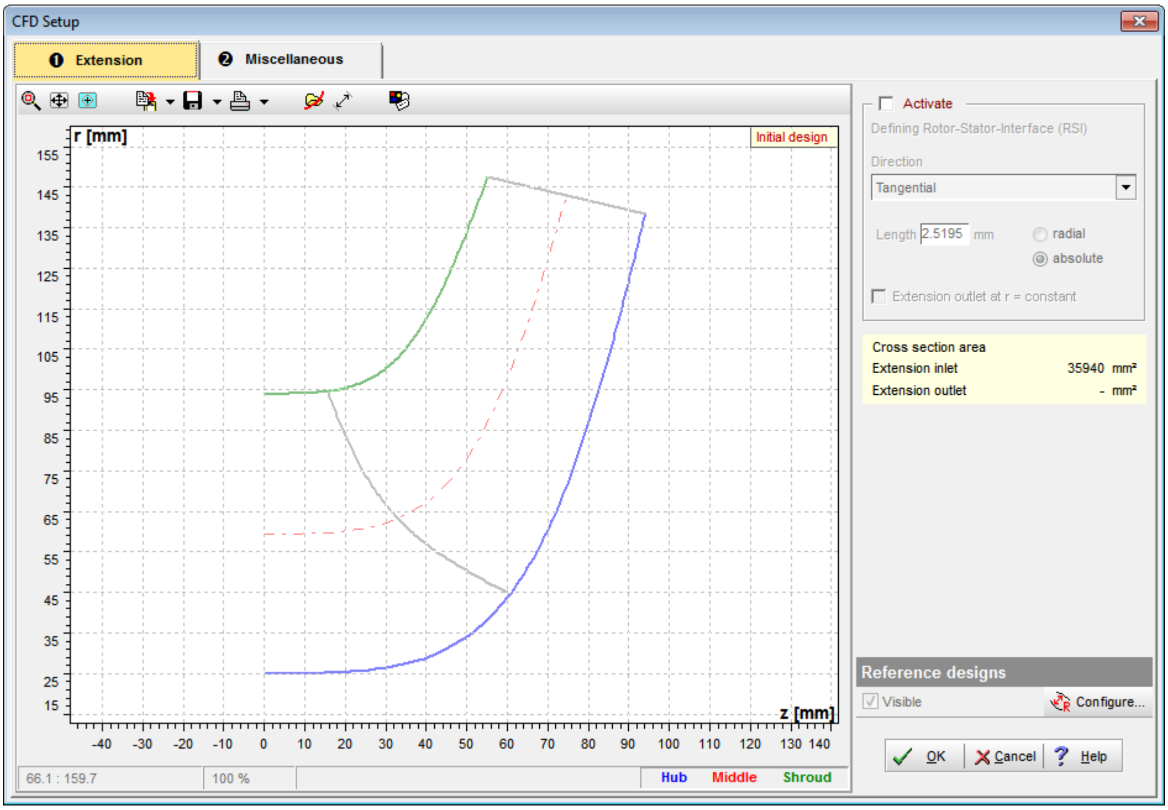
- Extension 368
- Impeller segment 371
- Blade O-Grid 375
- Through - flow area 376
- Blade projection 376

These extensions are to be used for flow simulation (CFD) and are virtual only.

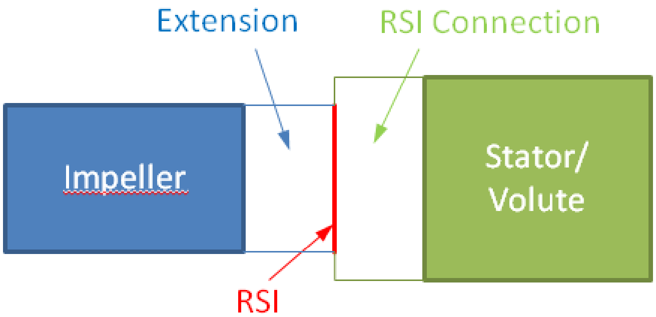
8.5.1 Extension

? Impeller | CFD setup | Extension

The designed geometry can be extended in meridional direction at the outlet.



The **extension** defines the Rotor-Stator-Interface (RSI). Its geometric parameters will be considered at the next component as inlet conditions so that the geometries as well as the meshes based on them match each other. Typically, the RSI is located in the middle of the rotating and the non-rotating component.



(RSI Connection: see Other 376)

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 $ß_2$.

The drop down menu **Direction** sets the direction of the extension. If it is set to tangential, hub and shroud will be tangentially extended.
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**, which means that the outlet of the extension is forced to be horizontal in the diagram (parallel to the z-axis).

The designed outlet extension will be displayed in the diagram automatically.

For unvaned stators, the extension is not necessary and therefore not activatable.

Possible warnings

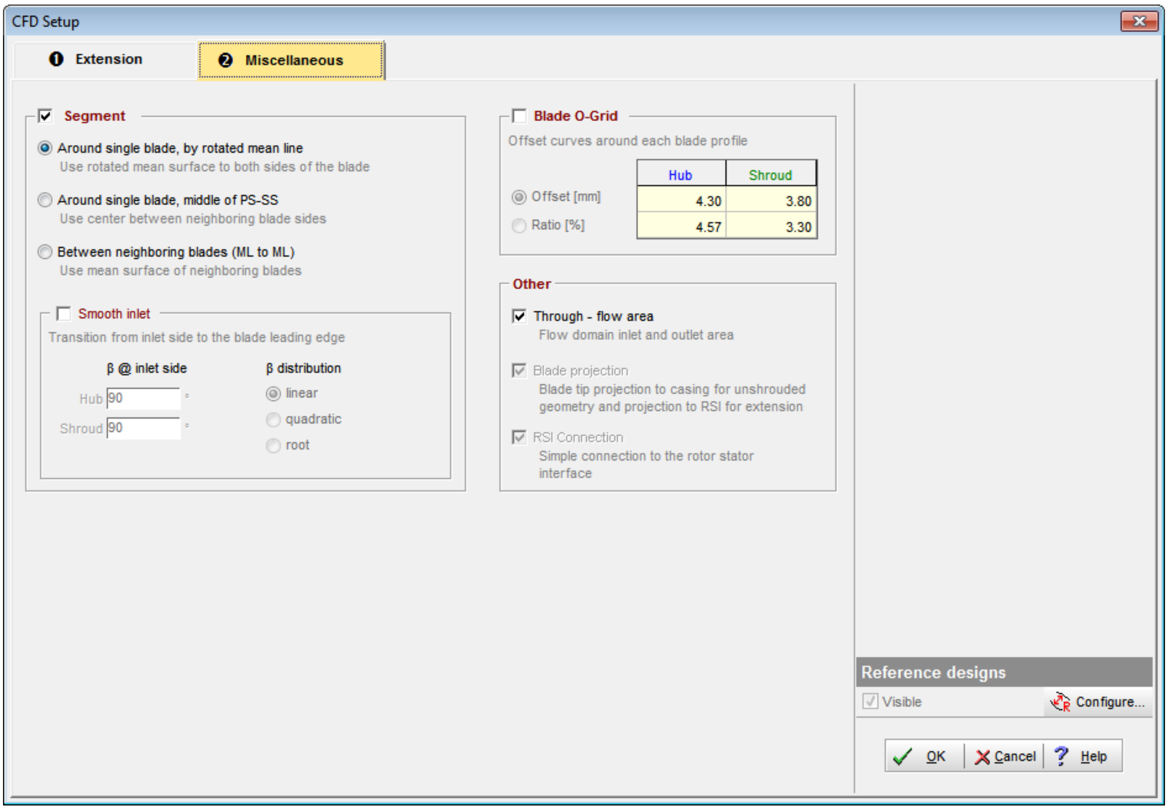
Problem	Lösungsmöglichkeiten
The length of the extension is smaller or equal to the distance tolerance. This might cause sewing defects in "Meridian.Flow Domain" during model finishing.	
The length of the extension is smaller or equal to the distance tolerance 376. This might cause geometrical defects when sewing faces during Model finishing 378.	If geometrical problems occur, change the distance tolerance or the length of the extension.
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.

8.5.2 Miscellaneous

? Impeller | CFD Setup | Miscellaneous

Miscellaneous virtual elements can be created:

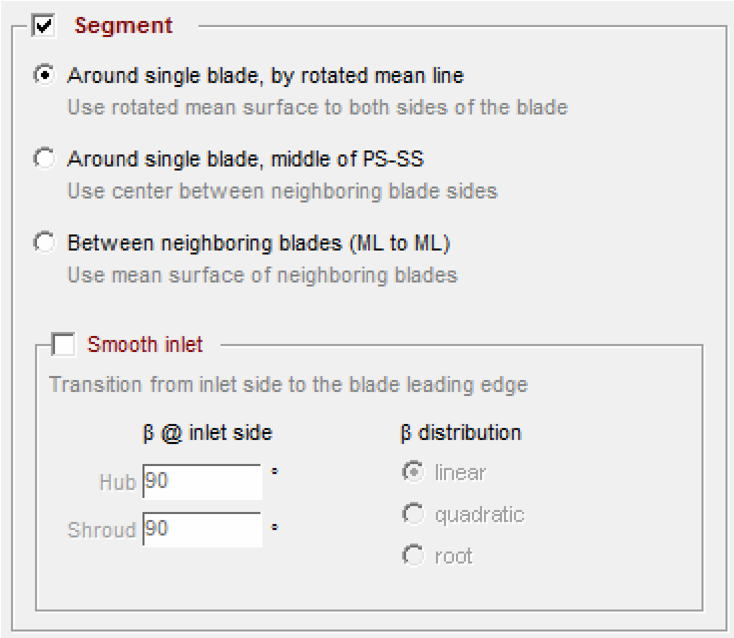
- Segment 371
- Blade O-Grid 375
- Other 376



8.5.2.1 Segment

? Impeller | CFD Setup | Miscellaneous | Segment

The **segment** is the flow passage around a single blade and represents the smallest rotation-symmetric part of the impeller.



There are the following options for the design:

"Around single blade, by rotated mean line"

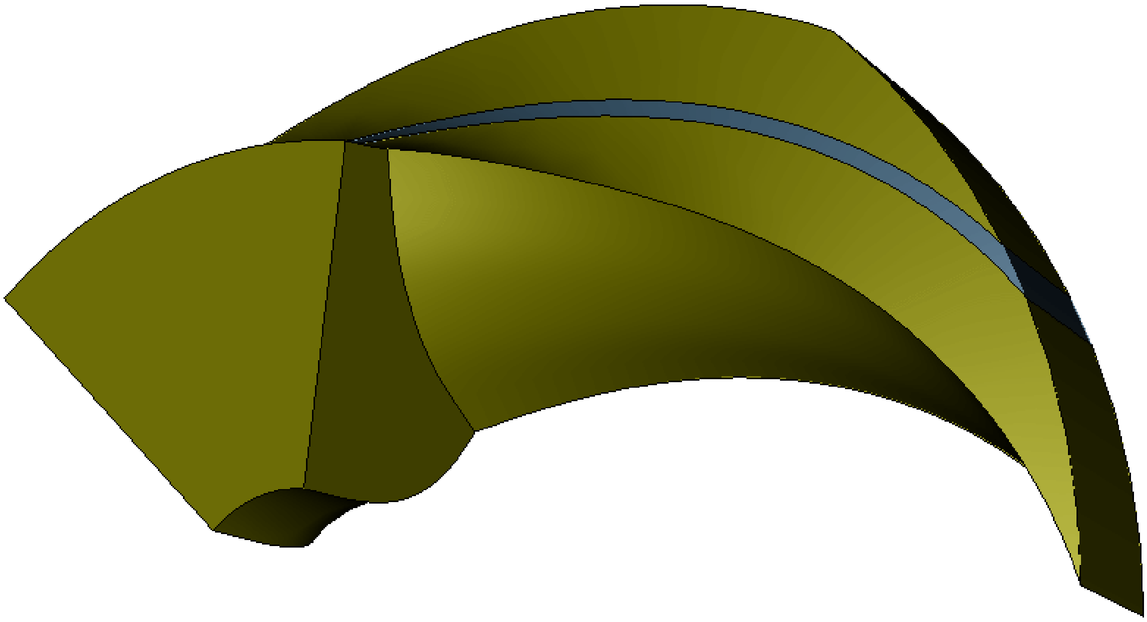
The blade mean surface is rotated to both sides to the middle of the flow channel.

"Around single blade, middle of PS-SS"

The middle of blade pressure and suction side of two neighboring main blades forms the segment boundary. This type should be used for thick asymmetric blades. It ensures that the blades do not cut the periodic surfaces of the segment.

"Between neighboring blades (ML to ML)"

The mean surfaces of two neighboring main blades form the segment boundary. This type is currently not supported by Model finishing 378.



"Around single blade, by rotated mean line"

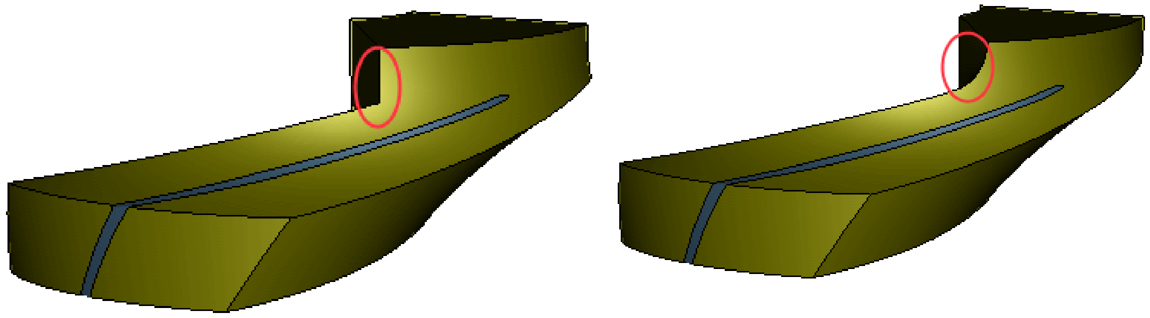
With **"Smooth inlet"** a smooth transition from the impeller inlet to the blade area can be designed. This surface is created by a virtual extension of the Blade mean line 319 from the blade leading edge (which is the trailing edge in case of turbines) to the Inlet 376 (Outlet for turbines).

There are three types of **β distribution** from the leading edge of the blade (**β_{B1}**) to the Inlet 376 (**β_{inlet}**):

- linear
- quadratic
- root

The values of β_{B1} have been defined in the Blade properties 307. At the Inlet 376 the distribution of the angle **β_{inlet}** is linear from hub to shroud.

without "Smooth inlet" with "Smooth inlet"



3D Model

The segment can consist of up to 3 solids:

- Segment.Real Geometry
  Segment of the flow passage bounded by real geometries (defined by Meridional contour 268)
- Segment.Extension
  Segment of the virtual geometry Extension 368 (optional)
- Segment.RSI Connection
  Segment of the virtual geometry RSI Connection 376 (optional)

Possible warnings

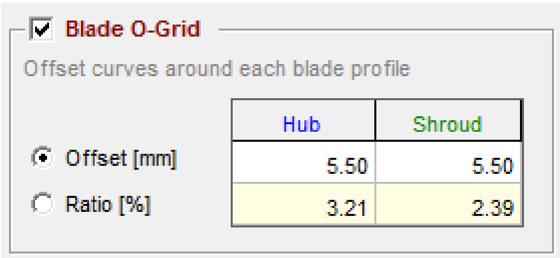
Problem	Possible solutions
Segment type "Around single blade, middle of PS-SS" is not applicable because the blade exceeds the meridional boundaries.	
This type of segment is incorrect if the blade exceeds the meridional boundaries.	Modify the blade so that it does not exceed the meridional boundaries or choose another type of segment.
3D-Error: Could not create solid for ... RSI Connection	
Unsupported RSI Connection geometry, e.g. only on one side (hub or shroud)	Uncheck RSI Connection 376 or change its geometry

Problem	Possible solutions
General solid creation problem	See 3D Model 183

8.5.2.2 Blade O-Grid

? Impeller | CFD setup | Miscellaneous | Blade O-Grid

Auxiliary curves for meshing can be designed that have a constant distance to the blade at each span.



Offset

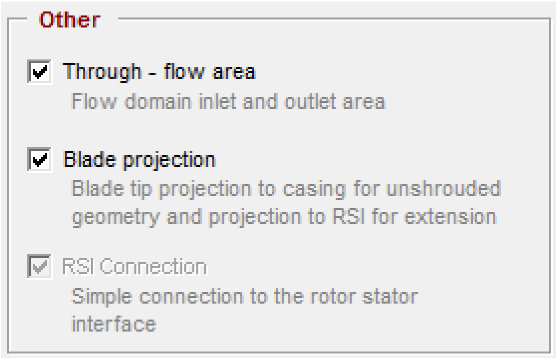
Absolute distance from the auxiliary curves to the blade

Ratio

Ratio of offset to the distance between neighboring blades (at blade center)

8.5.2.3 Other

? Impeller | CFD Setup | Miscellaneous | Other



Through-flow area

Inlet and outlet area define the inflow and outflow boundary of the whole flow channel.

Blade projection

In case of an unshrouded impeller the outer blade profile is projected onto the casing.

If an Extension 368 exists, the blade trailing edge is projected onto the RSI.

This option must be enabled for a successful export to ICEM-CFD (ANSYS) 131.

RSI connection

If a Rotor-Stator-Interface (RSI) is existing on the inlet side of the component, an existing gap can be closed automatically by the RSI connection. These surfaces provide a simplified, closed volume model for flow simulation neglecting impeller side chambers or other casing parts.

(see also Extension 368)

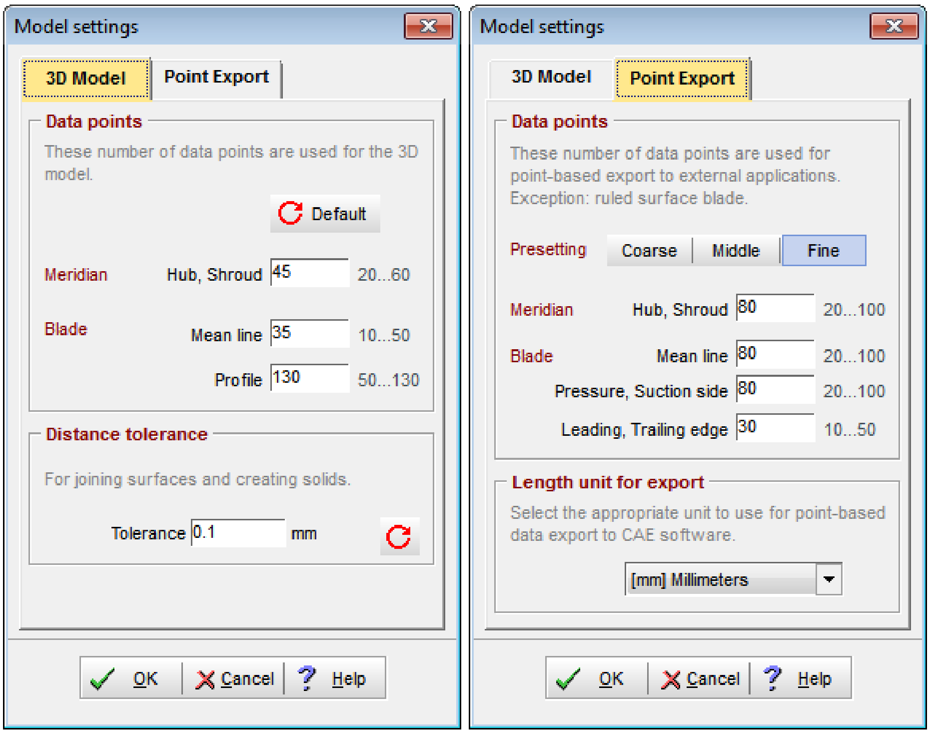
8.6 Model settings

? 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



3D Model

Distance tolerance (3D Model)

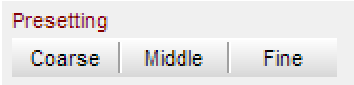
The distance tolerance defines the maximum allowed distance between sewed surfaces, e.g the faces of a solid.

If it is too small, the solids cannot be created.

If it is too big, small faces are ignored when creating a solid.

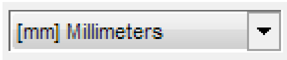
Point Export

Presetting



Select from 3 global presets.

Length unit for Export



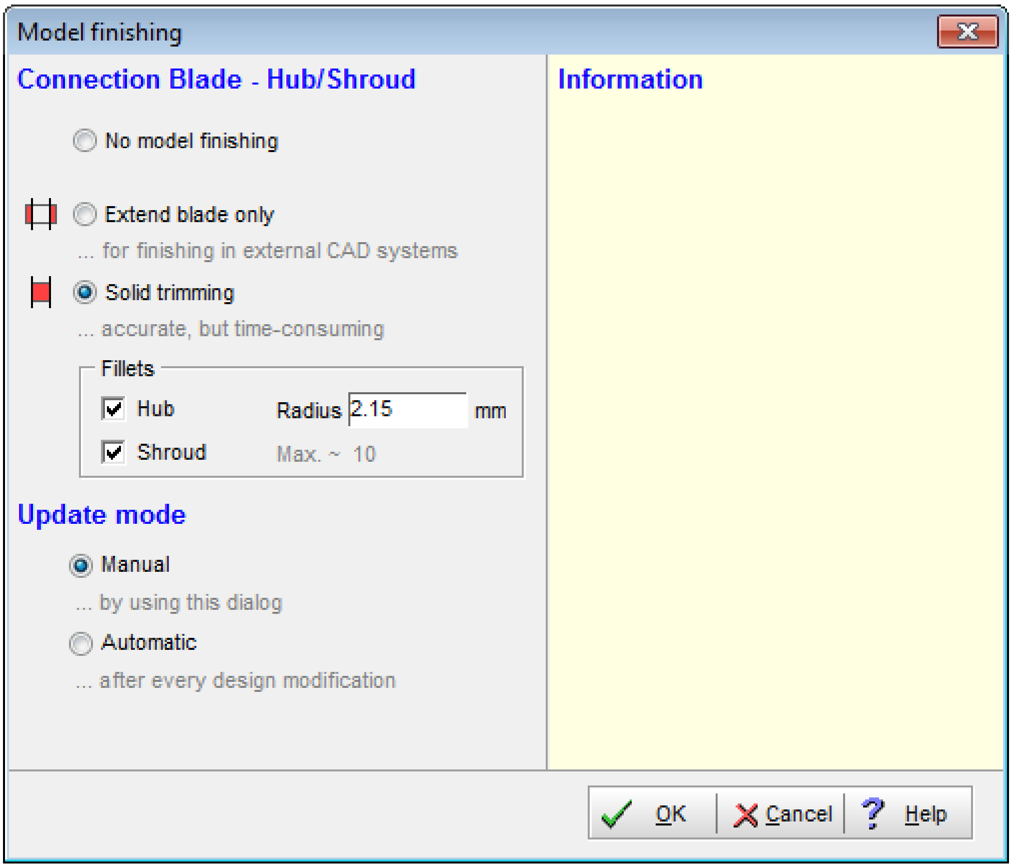
The length unit for the geometry export can be selected. Please select the appropriate units when importing data to the chosen CAD software.

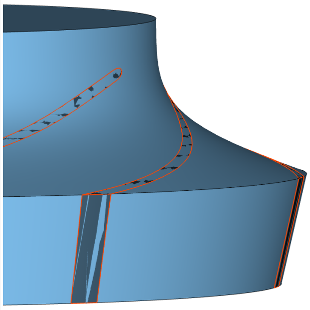
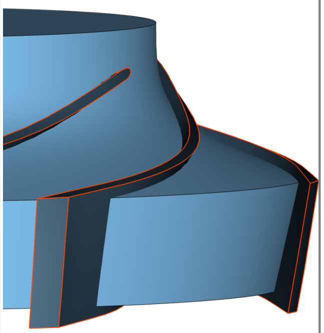
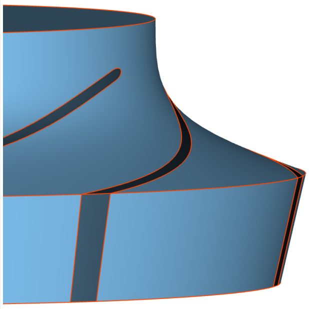
When a **new impeller** is created the model settings of the last opened impeller are carried over.

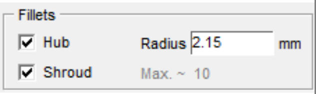
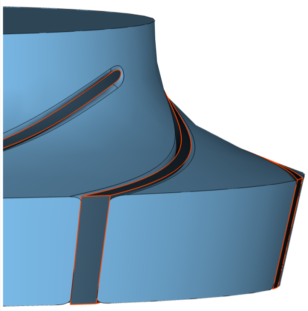
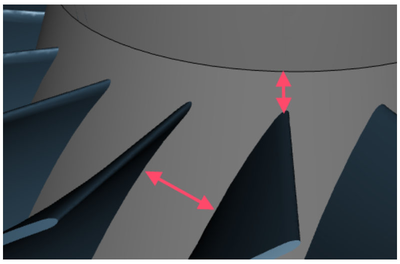
8.7 Model finishing

? Impeller | Model finishing 

The dialog offers different possibilities to design the connection between blade, hub and shroud.



No model finishing 	
Extend blade only 	Extends blades through hub, shroud and trailing edge; for later trimming in a CAD-system
Solid trimming 	Trims blade on hub, shroud and trailing edge; affects only the solids (and solid faces) of *Meridian.Flow Domain*, *Segment* and *Blade.* Trimming is only possible if the solids of *Meridian.Flow Domain* and *Blade* could be created successfully. Trimming is a **time-consuming** operation (up to 1 minute or some minutes for impellers with splitter blades). Because only solids are trimmed, **point-based exports cannot take advantage of this operation**. **Details:** *Solid trimming* is based on a Segment 371. If no segment is defined, it is created temporarily, not visible to the user.

	Internal workflow: • The blades are extended (see Extend blade only 379) • 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 368 or RSI Connection 376, *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*. • CFD Setup option: If Blade projection 376 was chosen, the corresponding projection surfaces are exactly trimmed.
Option: Blade root fillet  	Fillet creation at blade root; affects only the solids (and solid faces) of *Meridian.Flow Domain* and *Segment.* The **fillet radius** should not be larger than the recommended value.  Fillet creation is not possible if the fillets of two neighboring blades would meet or if the fillet would protrude beyond the impeller inlet.

Update mode

- Manual — The 3D-model is updated only after closing the dialog.
- Automatic — The 3D-model is updated after every design modification automatically.

Symbol in main window

The symbol shows the state of Model-finishing.

Model finishing is not defined yet.

The 3D-Model has been updated according to the finishing settings.

The design has been changed but the 3D-Model is **not up to date** (not finished) or the model finishing has failed.

Possible warnings

Problem	Possible solutions
Model finishing currently NOT up-to-date	
Model finishing was not executed yet; therefore the 3D model is not up-to-date	Open Model finishing 378 and click <OK>
Extend/solid trimming could fail due to high tangential difference between hub and shroud at leading/trailing edge and low number of spans.	
Very low number of spans	Increase the number of spans 307 up to at least 4
Finishing type was reset to "No model finishing". Solid trimming is not supported for the selected segment type "Between neighboring blades (ML to ML)". See CFD Setup/ Miscellaneous/ Segment.	
Solid trimming is not supported for "ML to ML" segment type.	Change segment type 370
Finishing type was reset to "No model finishing". Solid trimming is not possible.	
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 344
Fillets are not supported.	
Fillets are not supported if solid trimming is not possible.	-

Problem	Possible solutions
Fillets creation on shroud was deactivated.	
Fillets on shroud are not supported for unshrouded designs.	-
3D-Error: Finishing failed!	
Leading edge very near to inlet	Change Meridional contour 268: Move leading edge towards outlet
Inlet (nearly) tangential to hub or shroud	Change Meridional contour 268: Avoid tangentiality
3D-Error: Finishing failed! (Fusing solids)	
Fusing of real geometry with CFD Setup components (Extension or RSI Connection) failed.	Increase the number of spans 307 or Remove Extension / RSI Connection from CFD Setup 368
3D-Error: Blade projection to RSI failed!	
Projection of blade to RSI (Extension) failed.	Change CFD Setup 368: Modify Extension or remove Blade projection
3D-Error: Blade tip projection to casing failed!	
Projection of blade to casing (shroud) failed.	Change CFD Setup 370: Remove Blade projection or RSI Connection

Part IX

9 Stator

? Stator

This chapter describes in detail the design process for stator type components featured in CFturbo.

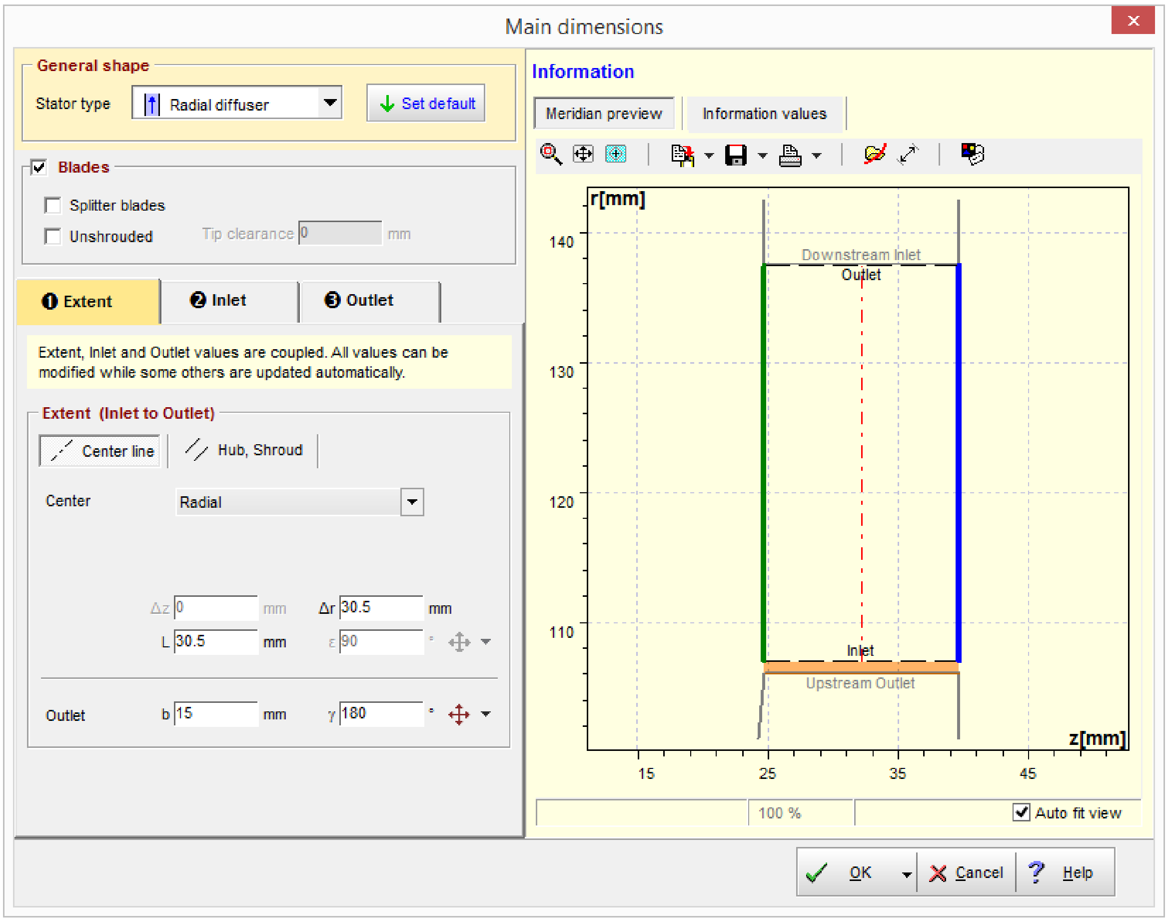
The content reflects the design steps in the sequence they are encountered during the design process.

Design steps

9.1 Main dimensions

? Stator | Main dimensions

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



General Shape

Here you can define the stator type initially. Currently the following types are available:

- Free form
- Radial diffuser

Using the button "Set default" you can set default properties for each stator type.

Blades

Here you can define if the stator should be vaned or unvaned.

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**.

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 387, Inlet 389 or Outlet 390.
  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.

Details

→ Number of blades 392

→ Extent 387

→ Inlet 389

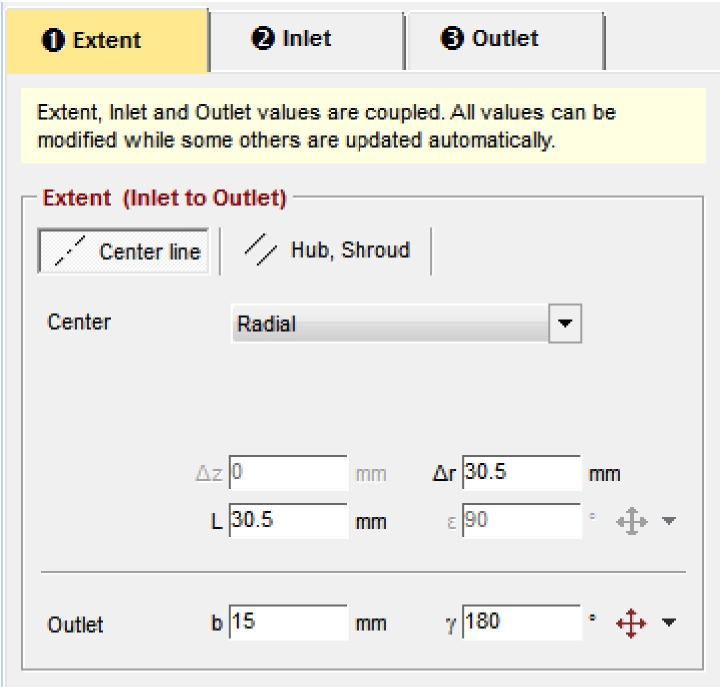
→ Outlet 390

Possible warnings

Problem	Possible solution
Hub/ Shroud/ Midline length is 0 (unrealistic geometry).	
The extent 387 of the stator is 0 at hub, shroud or midline.	Specify a reasonable length value or remove the stator completely.

9.1.1 Extent

Stator extent has to be considered in relation to its inlet 389 and outlet 390. 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

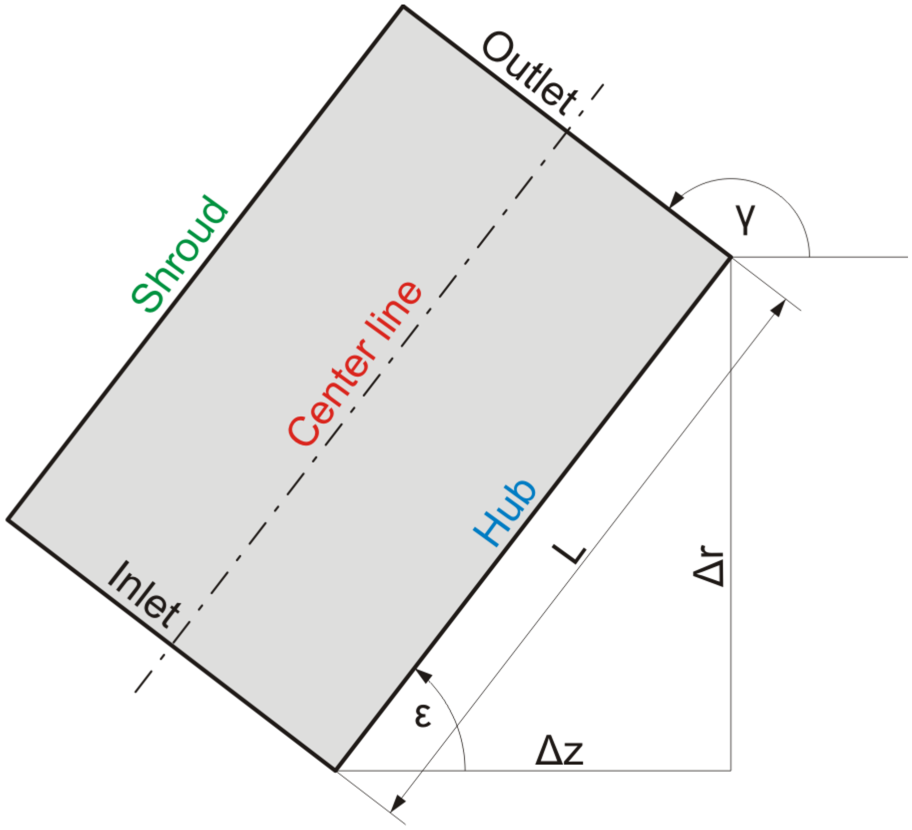
- preselection of extension direction:Radial, Axial, Tangential (to outlet of previous component), Free form
- Definition of axial extension **Δz** and radial extension **Δr**
or
length **L** and angle of center line to horizontal direction **ε**
- Definition of end cross section (Inlet or Outlet) by width **b** and angle to horizontal direction **γ**

2. Hub, Shroud

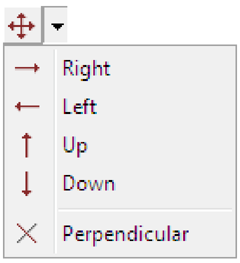
separately for hub and shroud:

- 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 **ε**



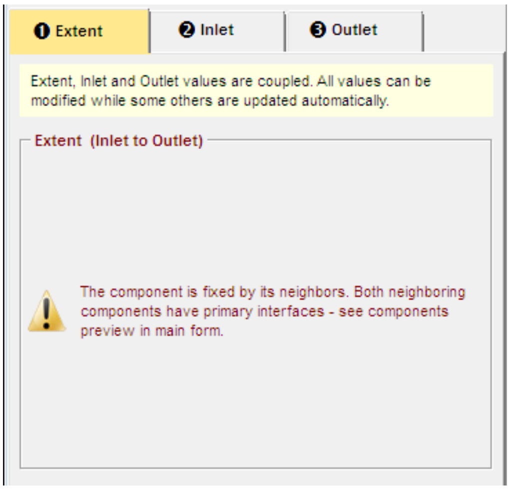
The angles ε and γ are defined by 0° horizontal right and rising in counter clockwise direction (mathematical positive). A menu with some default angles is supporting angle input:



0°	
180°	
90°	
270°	
Perpendicular:	ε perpendicular to inlet or outlet cross section
Parallel:	γ parallel to inlet or outlet cross section

Depending on the interface[38] type the extents are defining the inlet or the outlet of the component.

If the stator has the primary interface side at outlet the extents will modify the outlet. Otherwise if the stator has the primary interface side at inlet then the inlet will be defined by the extents.



If the neighboring components are primary both at inlet and at outlet then the extent of the stator cannot be specified explicitly because it's clearly defined by these interfaces.

Information

Design point	Design point information, see Global setup 71
Ratio outlet to inlet	
Diameter ratio	d_{Out}/d_{In}
Width ratio	b_{Out}/b_{In}
Area ratio	A_{Out}/A_{In}
Inlet area	A_{In}
Outlet area	A_{Out}

9.1.2 Inlet

Here you can define the inlet of the stator.

If the outlet can be modified then it's updated by addition of extent to inlet. Otherwise the extent will be adapted.

→ Details: see Interface definition 40

9.1.3 Outlet

Here you can define the outlet of the stator.

If the inlet can be modified then it's updated by subtraction of extent from outlet. Otherwise the extent will be adapted.

➜ Details: see Interface definition 40

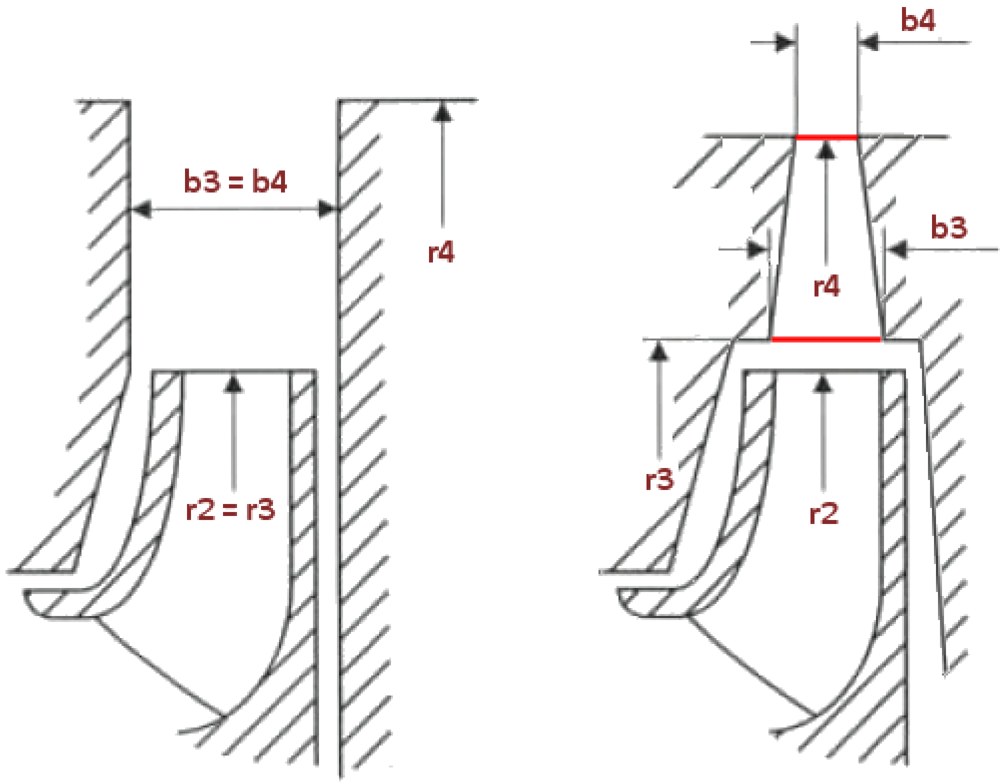
9.2 Meridional contour

? Stator | Meridional contour

In principle, the same features are available as for the meridional design 268 of impellers.

The endpoints of hub and shroud curves are fixed by main dimensions 384 and cannot be modified here.

For "Radial diffuser" type of stators (see main dimensions 384) the following geometrical dimensions are defined:



9.3 Blade properties

? Stator | Blade properties

In principle, the same features are available as for the blade properties 292 of impellers.

To support the selection of a suitable blade count a separate dialog 392 can be used, which can be started by pressing the button right beside the edit field.

The outlet angles αTE are input values for most of the blade types according to the desired change of flow direction. Slip models are not available for stators. Some angle oversizing should be considered if necessary.

Two additional special **blade shapes** are available for "Radial diffuser" type stators (see Main dimensions 384):

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 394).

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 r3, inlet angle α3 and ideal throat width a3.

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 394).

Calculation of throat width a3 can be done using the conservation of angular momentum (const. swirl) or a specific deceleration ratio alternatively:

a) Const. swirl

Throat width corrsponds 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).

b) Deceleration

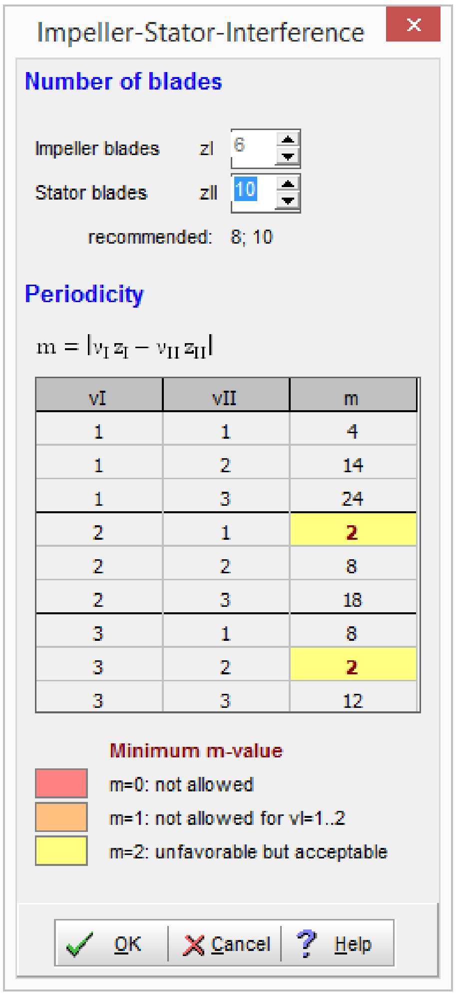
Alternatively one can use the deceleration ratio c_{3q}/c_2 (0.7...0.85) for throat width calculation.

$$a_3 = \frac{Q}{z b_3 c_2} \cdot \left(\frac{c_2}{c_{3q}} \right)$$

Trailing edge angle αTE is a result of mean line design for these special blade shapes and therefore cannot be specified explicitly ("**var.**").

9.3.1 Number of blades

Number of blades, stator outlet diameter and minimum blade distance are significant for the actual diffuser part of the stator and therefore have high influence on the flow losses. These 3 parameters have to be adjusted carefully.



The number of blades of impeller and stator has to be coordinated carefully in order to minimize pressure pulsation and therefore mechanical load and noise emission.

The number of impeller blades is defined and fixed by 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 $p_I = \nu_I \cdot z_I$
- stator periodicity $p_{II} = \nu_{II} \cdot z_{II}$

  (ν = 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:

$$m = \left|p_I - p_{II}\right| = \left|\nu_I \cdot z_I - \nu_{II} \cdot z_{II}\right|$$

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 (ν_I=1; ν_I=2) due to unacceptable shaft vibration, if possible also in third order (ν_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 (ν_I = 1..3) and (ν_{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.

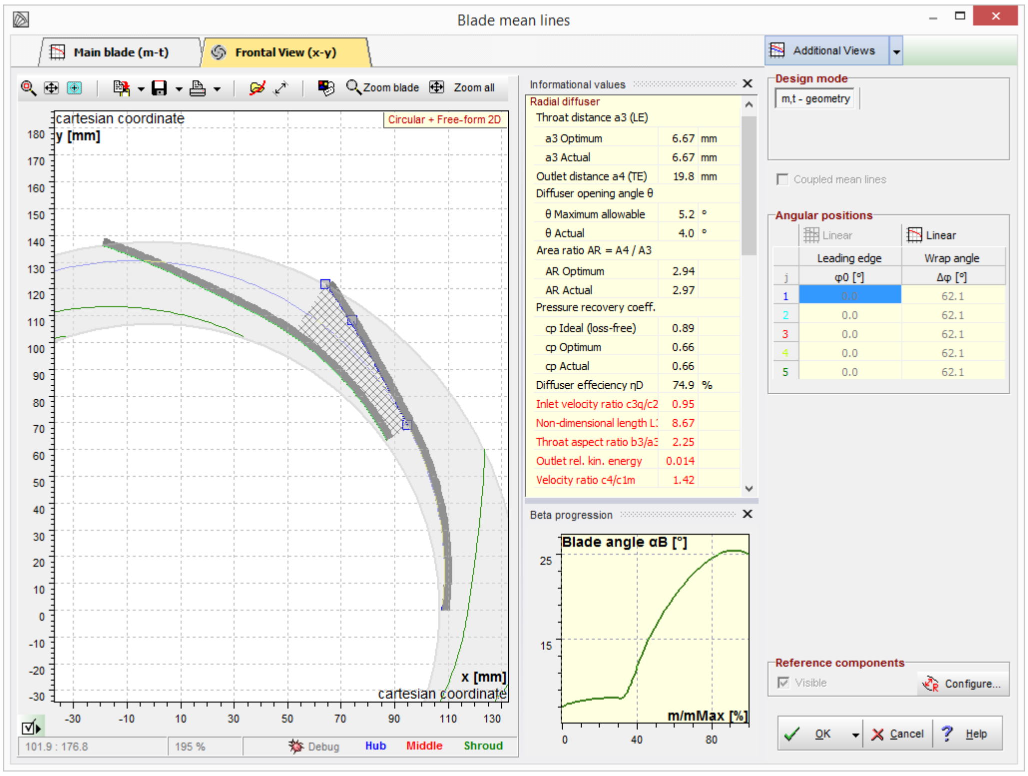
The recommended stator blade count according to the current number of impeller blades are represented below the input field.

9.4 Blade mean lines

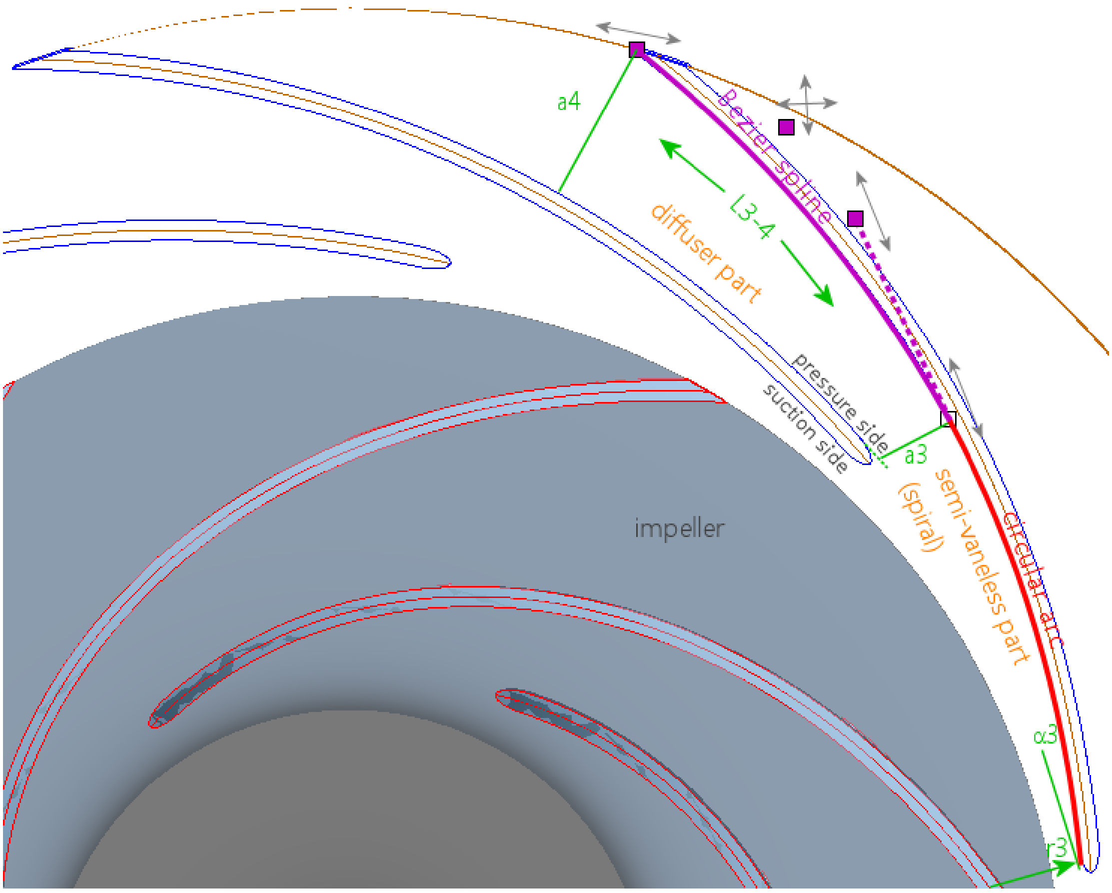
? Stator | Blade mean lines

In principle, the same features are available as for the mean lines 319 of impellers.

For special radial diffuser blade shapes "**Log. Spiral + Straight 2D**" and "**Circular + Free-form 2D**" the mean line design is made in the frontal view. The mean lines are the inner vane sides (concave sides).



Initially the blade thickness is ignored for the mean line design (red/magenta in the sketch). The opposite side if the flow channel is generated by rotation and adding the blade thickness. The blade thickness is assumed as linear between sLE and sTE (see blade properties 391), if the thickness distribution was not defined yet. Otherwise the thickness distribution defined in the blade profile 397 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 a3 (LE)	Throat width at inlet (leading edge)	
a3 Optimum *	Optimal value: average of calculation by const. swirl and deceleration ratio	see blade properties 391
a3 Actual	Actual value: shortest distance from vane leading edge to neighboring vane	
Outlet distance a4	Shortest distance from vane trailing edge to neighboring vane	
Diffuser opening angle θ	Allowable diffusion angle	

θ Maximum allowable	Max. allowable value to avoid flow separation depending on equivalent inlet radius and length	$\vartheta_{max} = 16.5° \cdot \sqrt{\frac{R_{3,eq}}{L}}$ $R_{3,eq} = \sqrt{\frac{a_3 b_3}{\pi}}$
θ 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=A4/A3	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. cp	Pressure recovery of the diffuser identified by a dimensionless coefficient	$c_p = \frac{p_4 - p_3}{\frac{\rho}{2} c_3{}^2}$
cp Ideal (loss-free) *	Pressure recovery in an ideal (loss-free) diffuser	$c_{p,id} = 1 - \frac{1}{A_R{}^2}$
cp Optimum *	Pressure recovery for optimal area ratio A_R	$c_{p,opt} = 0.36 \left(\frac{L_{3-4}}{R_{3,eq}}\right)^{0.26}$
cp Actual *	Pressure recovery in real diffuser (with energy losses)	based on test results; plotted in diagrams; target: $c_{p,act} = c_{p,opt}$
Diffuser effeciency ηD *	Diffuser efficiency	$\eta_D = \frac{c_p}{c_{p,id}} = \frac{c_p}{1 - \frac{1}{A_R{}^2}}$
Inlet velocity ratio c3q/c2	Inlet deceleration ratio	$c_{3q}/c_2 = 0.7...0.85$ for low specific speed
Non-dimensional length L34/a3	Ratio of length to throat width	$L_{3-4}/a_3 = 2.5...6$

Throat aspect ratio b3/a3	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	$\frac{c_4^2}{2gH_{opt}} = 0.02 \ldots 0.04$
Velocity ratio c4/c1m *	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**

9.5 Blade profiles

? Stator | Blade profile

In principle, the same features are available as for the blade profiles 337 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 394).

For radial diffusers the same informational values as in the mean line design 394 are displayed in the Info area. The reason is the influence of the blade thickness to these numbers.

9.6 Blade edges

? Stator | Blade edges

In principle, the same features are available as for the blade edges 344 of impellers.

9.7 CFD Setup

? Stator | CFD Setup

In principle, the same features are available as for the CFD setup 368 of impellers.

9.8 Model settings

? Stator | Model settings

In principle, the same features are available as for the model settings 376 of impellers.

9.9 Model finishing

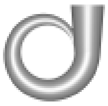
? Stator | Model finishing

In principle, the same features are available as for the model finishing 378 of impellers.

Part X

10 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

- Inlet definition 400
- Cross section 406
- Spiral development areas 417
- Diffuser 428
- Cut-water 434
- Model settings 445
- CFD setup 444

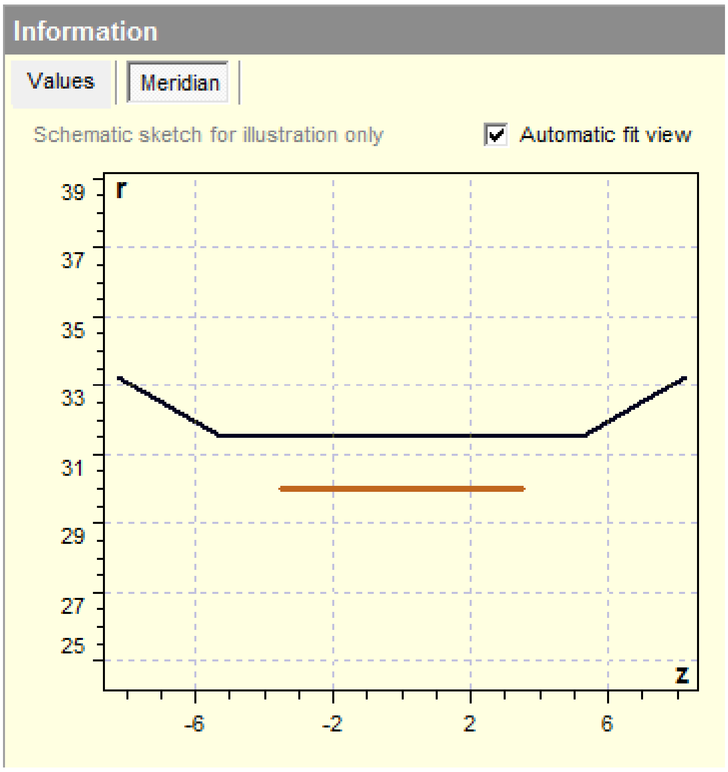
10.1 Setup & Inlet

? Volute | Setup + Inlet

The first design step of the volute is to define the inlet side. It consits of 2 steps:

(1) Setup 401

(2) Inlet details 405



On right panel **Information** on page **Meridian** 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.

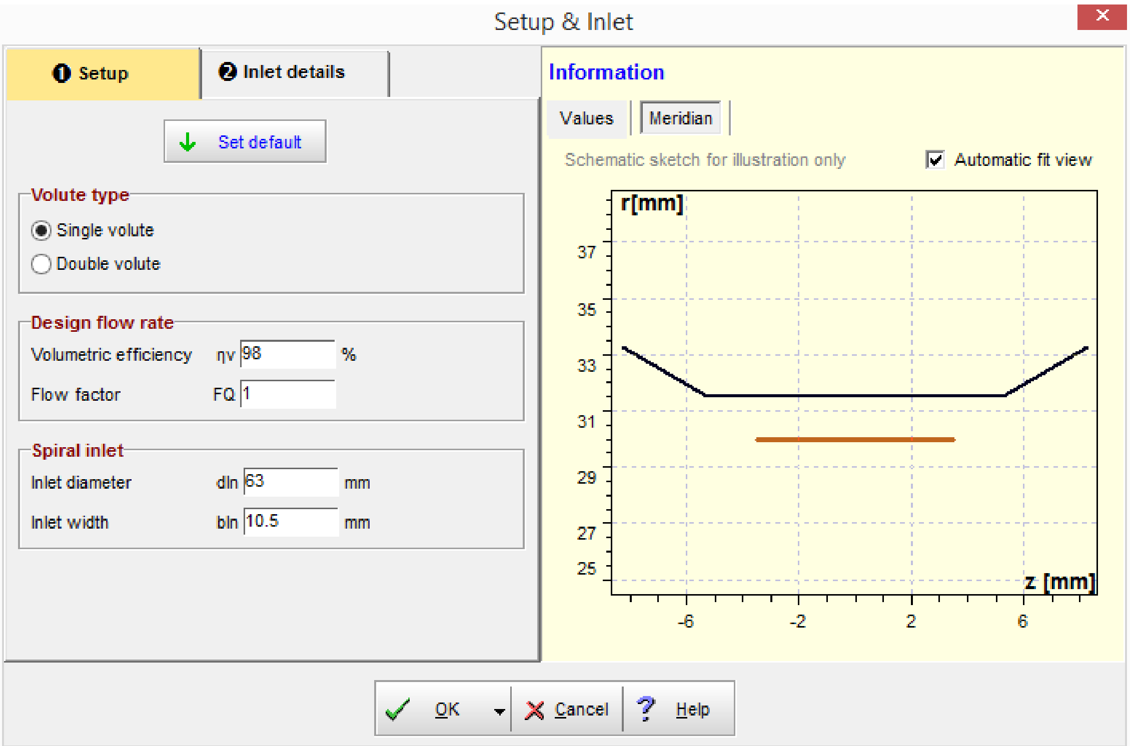
Auto fit view results in automatic scaling of the diagram if geometrical values are changing.

10.1.1 Setup

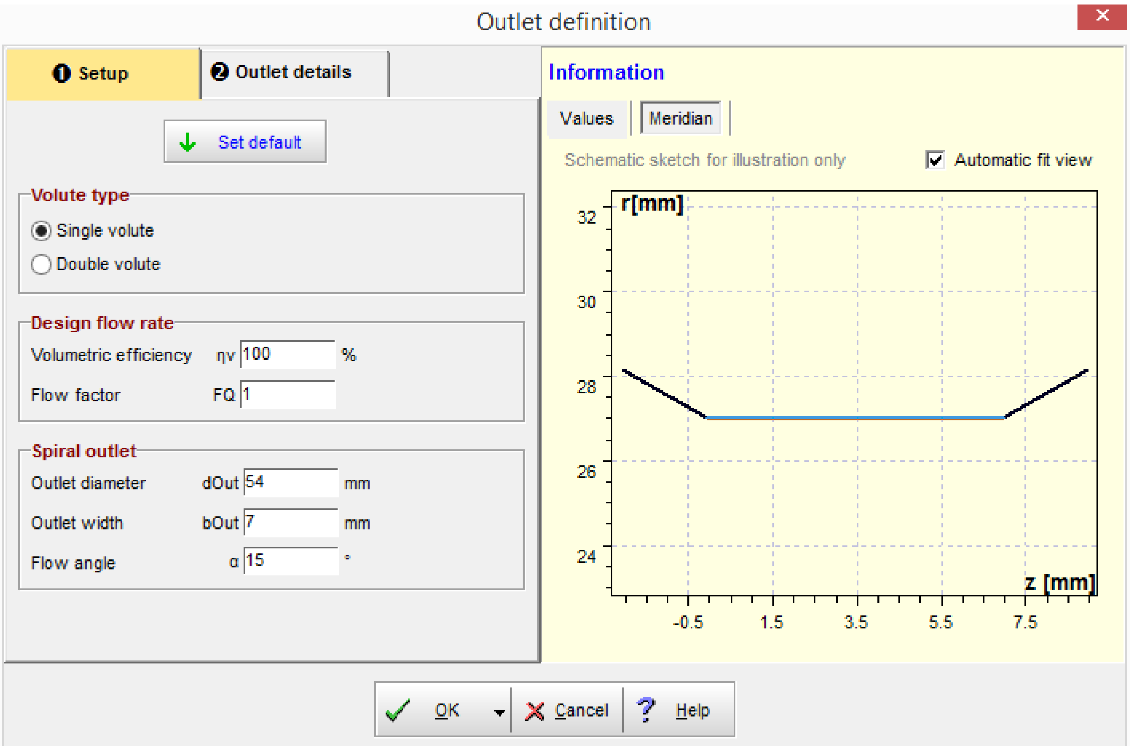
On page **Setup** you can define some general properties used for the spiral design.

Depending on the project type different input parameters are required (see below).

for pumps, ventilators, compressors



for turbines



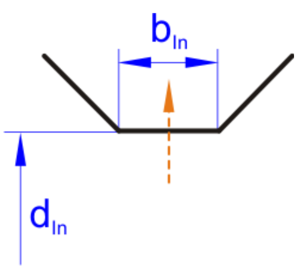
Volute type

- 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)

Please note:

For stand-alone volutes you have to define the inlet interface first, see Inlet Details 405, 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 145).

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 405.

[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**):

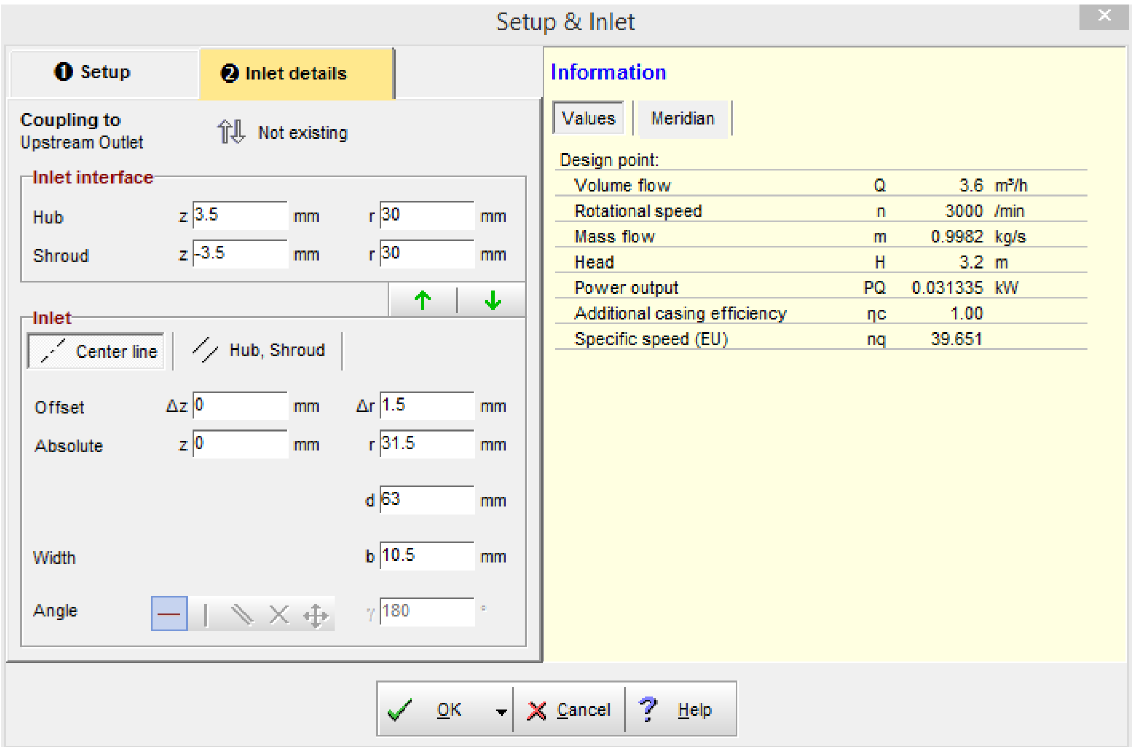
Calculated internal flow rate Q_i	$Q_i = F_Q \cdot Q/\eta_V$
Inlet/Outlet diameter ratio	d_{In}/d_2
Inlet/Outlet width ratio	b_{In}/b_2
Inlet/Outlet meridional velocity	c_m

Inlet/Outlet circumferential velocity	c_u
Inlet/Outlet velocity	c
Inlet/Outlet flow angle	α

10.1.2 Inlet details

On page **Inlet details** the details of the inlet interface can be specified.

Details: see Interface Definition 38



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 401.

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

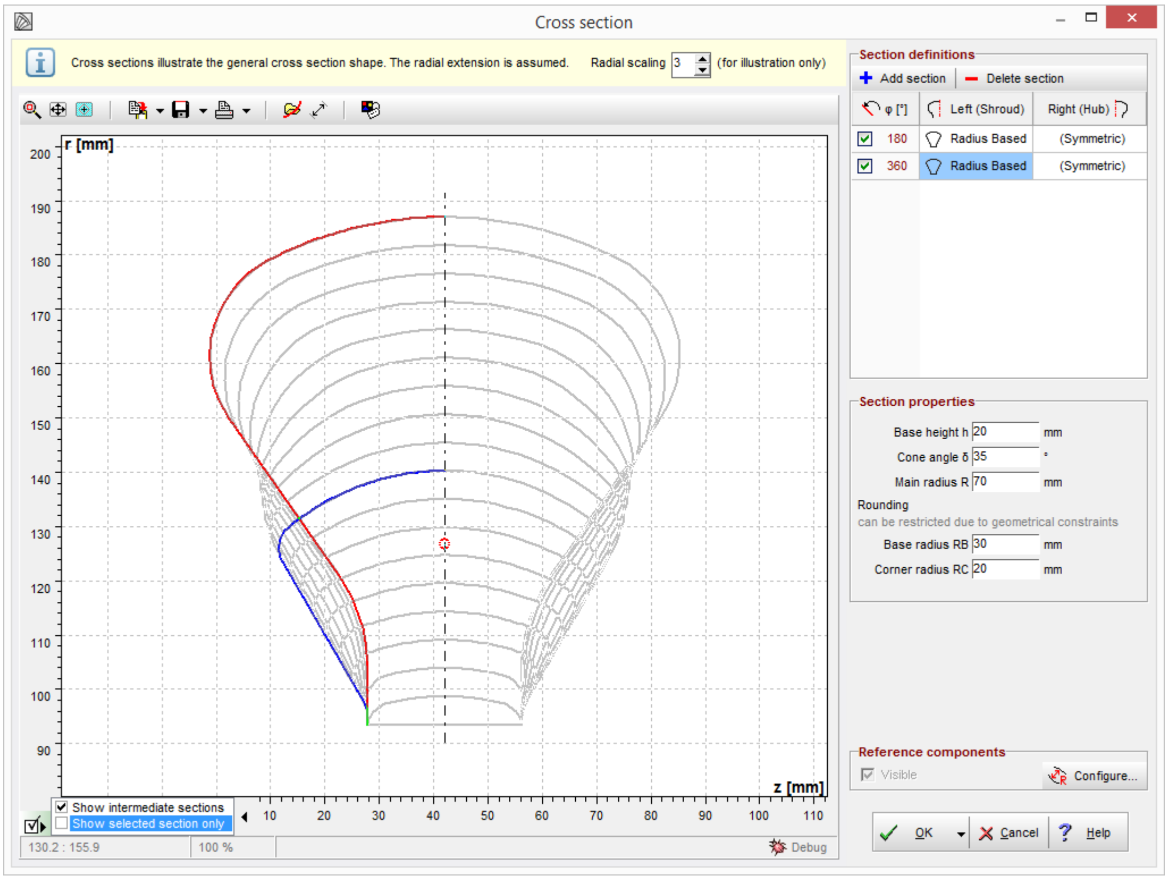
Right in the panel **Information** on page **Values** some values are displayed for information. These are values of the design point (Global setup 71) and flow properties on the outlet of upstream component.

10.2 Cross Section

? Volute | Cross Section

The shape of the cross-section of the volute can be selected here. The general cross section shape is illustrated whereas the radial extension is assumed (radial scaling can be modified above the diagram).

In general, very small cross-sections width should be avoided. The achievable cross-section shape strongly depends on manufacturing and the available space.



Sections

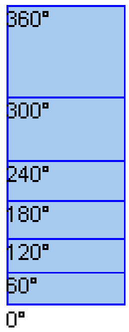
The table contains the cross section definitions (at least 1 cross section). Each cross section is defined by:

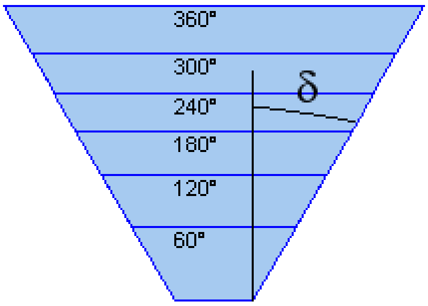
- the circumferential position: angle φ
- (de)activation by selecting the checkbox on the left side (at least 1 cross section has to be active)
- cross section type on the left side
- optional cross section type on the right side or symmetric

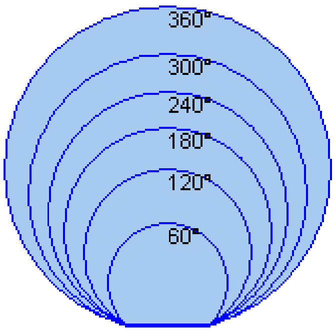
The section definition is running in the range 0° < φ ≤ 360°. The section at φ=0° is flat always - therefore a section definition at this position makes no sense.

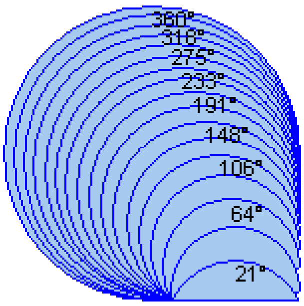
Between 2 neighboring cross section definitions a smooth transition is realized. If only a single section is defined then this definition is used for all circumferential positions.

The following cross section types are available:

<table>
<tr>
<td>
</td>
<td>Rectangular

most simple cross-section shape; cannot be achieved in cast parts; only sensible for low specific speeds, since otherwise the cross-section becomes too large</td>
</tr>
<tr>
<td>
</td>
<td>Trapezoid

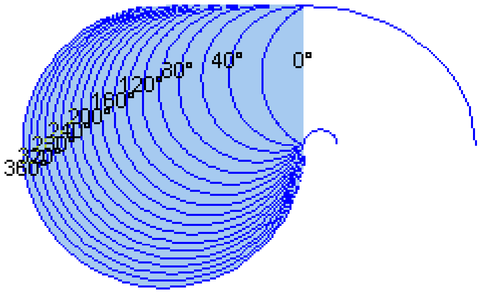
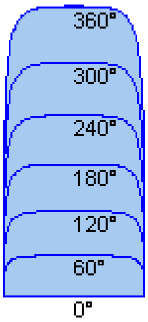
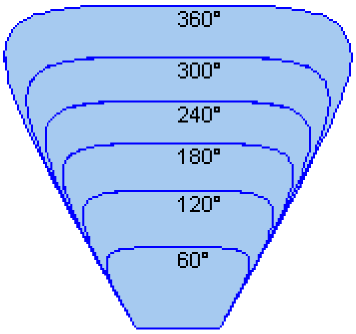
cannot be achieved in cast parts; the angle δ can be specified; results in a flatter cross-section than a rectangular cross-section, with less intense secondary flow</td>
</tr>
<tr>
<td>
</td>
<td>Round - symmetric

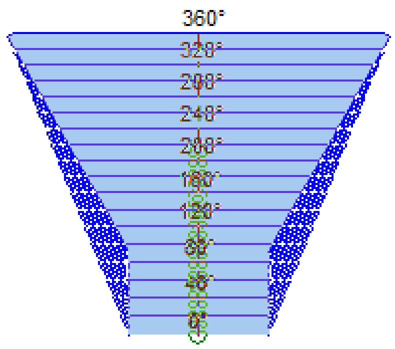
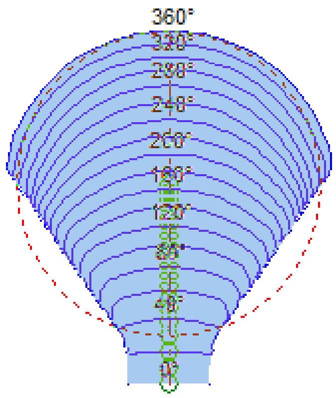
simple geometry with a beneficial stress distribution; does not develop on rotation surfaces</td>
</tr>
<tr>
<td>
</td>
<td>Round - asymmetric, external

more favorable secondary flow structure than with a symmetrical cross-section; often with mixed-flow impellers

• Strictly external: cross sections don't fall below inlet radius

• Open to right: asymmetric development</td>
</tr>
</table>

	to right (pos. z-direction) • **Square on top**: square shape on right top of cross section
	**Round - asymmetric, internal** limitation of radial extension; additional bend necessary see Internal cross sections 416
	**Bezier - Rectangle type** analogous with **Rectangle**; with chamfers (cast radii) see Bezier cross section 411
	**Bezier - Trapezoid type** analogous with **Trapezoid**; with chamfers (cast radii) see Bezier cross section 411

	**Line segments** see Line Segments cross section 412
	**Radius based** see Radius based cross section 415

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.

Display options

Under **Display options**, changes can be made which affect only the graphics.

Limitations

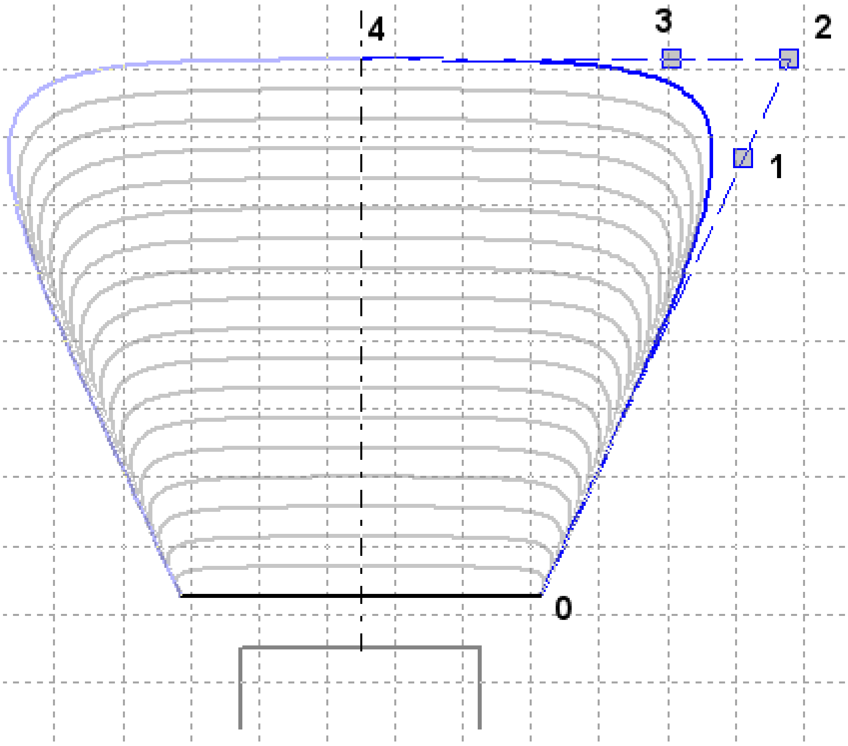
For double volutes the cone angle (opening) of all cross sections has to be constant. Therefore round types and Line segments are not available.

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 400). 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

10.2.1 Bezier cross section

The shape of a **Bezier** cross-section is described by a Bezier curve.



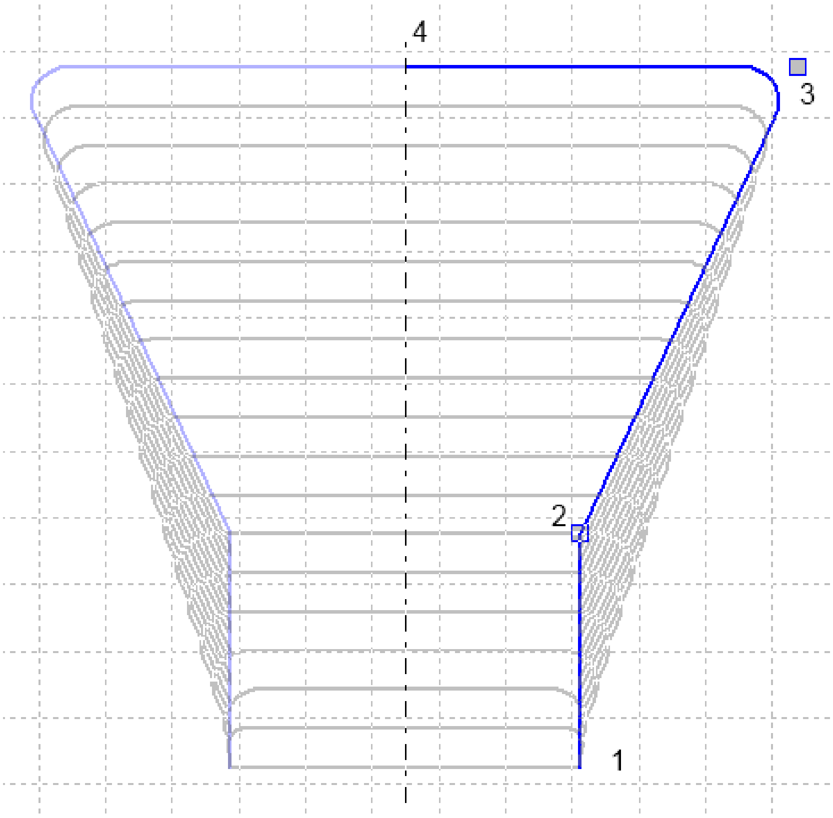
One half of the shape of the cross-section is described using a 4th degree Bezier polynomial. Points 0 and 4 are the end points and cannot be changed. Point 1 can be moved along a straight line which corresponds to the cone angle of the cross-section (0° for a rectangle type, δ for a trapezoid type). 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.

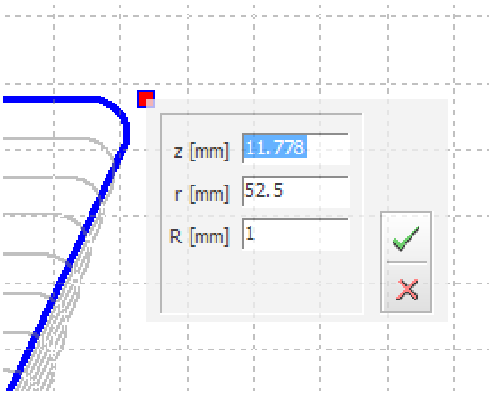
10.2.2 Line Segments cross section

The shape of a **Line segments** cross-section is described by a series of line segments.



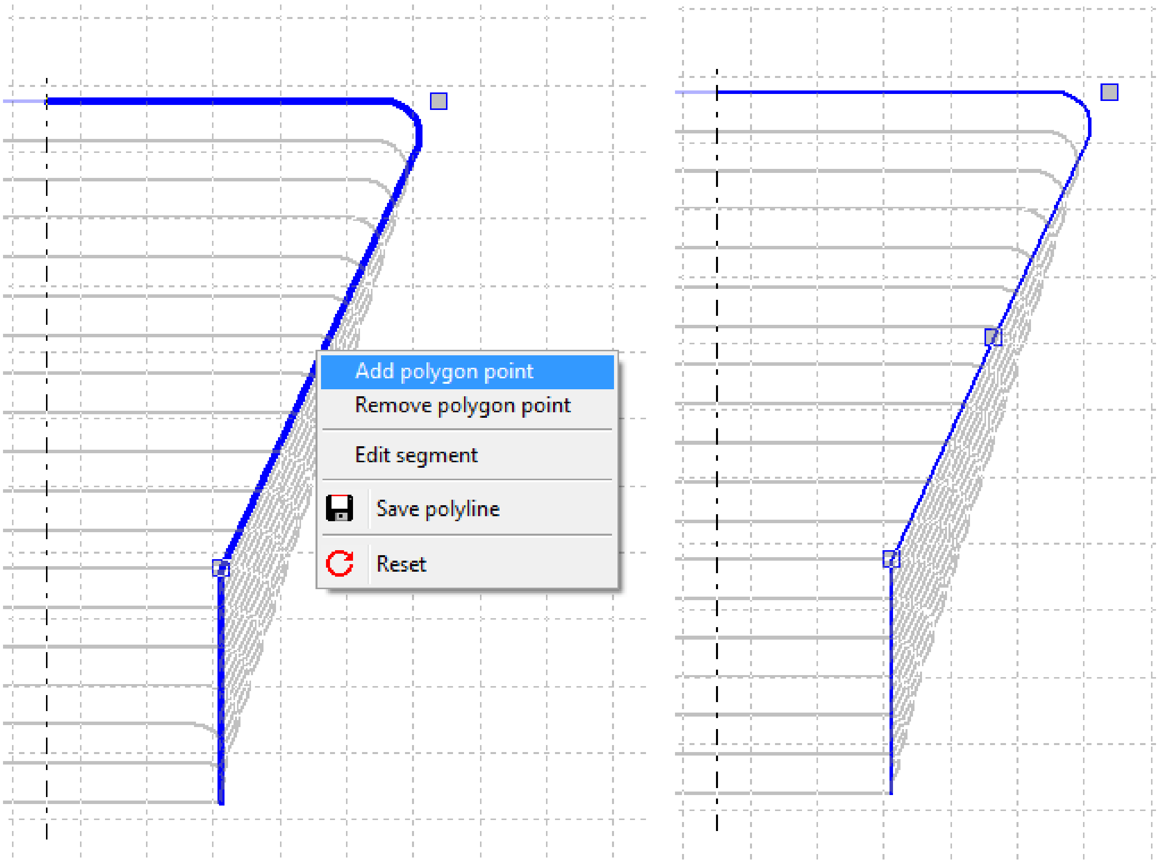
One half of the shape of the cross-section is initially based on line segments arranged in a trapezoid shape. Points 1 and 4 are the fix start- and endpoint.

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:

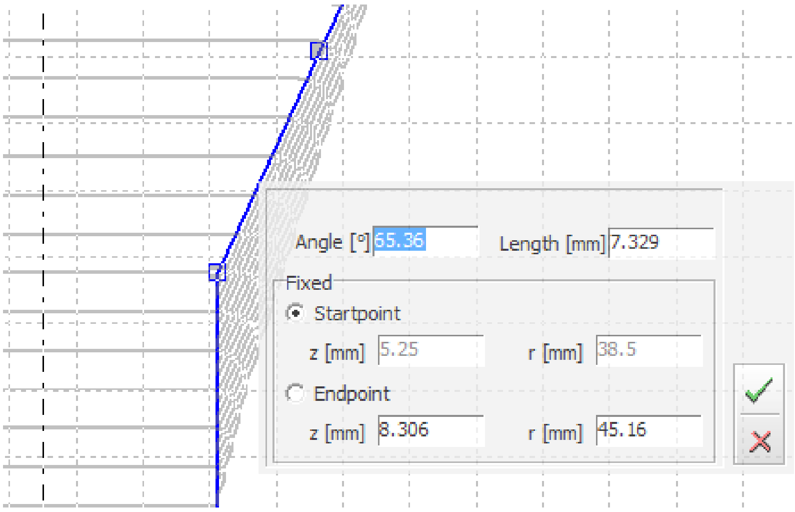


Coordinates and radius of vertex

Using the context menu of a line segment, points can be added at the cursor position or be removed:



The context menu also offers to display and edit the values of the segment. Either the start- or endpoint of the segment can be changed. In some cases, like in sample 1, the segment between point 1 and 2 has a fixed start point according to the geometrical constraints.

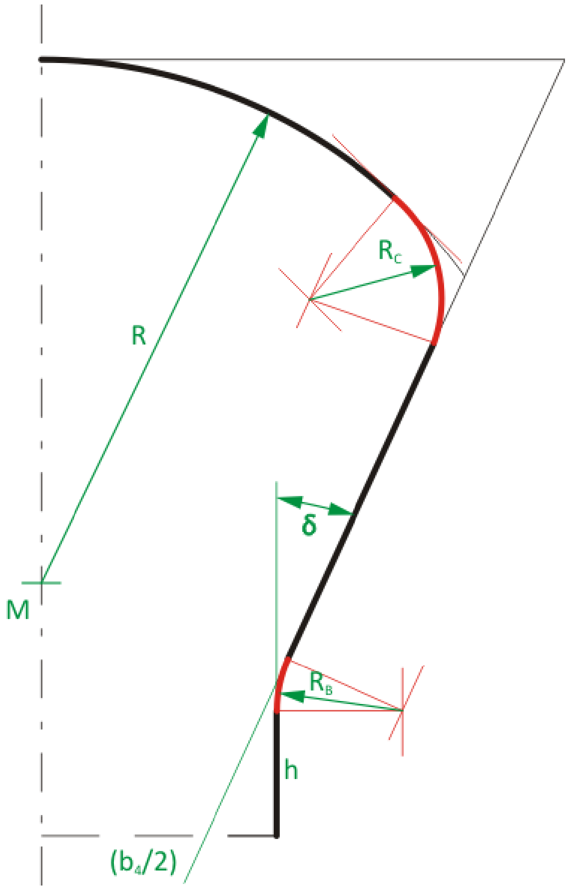


When moving points the following constraints can be enforced by pressing a key on keyboard:

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.

10.2.3 Radius based cross section

The shape of a **radius based** cross section is described by straight lines and circular arcs.

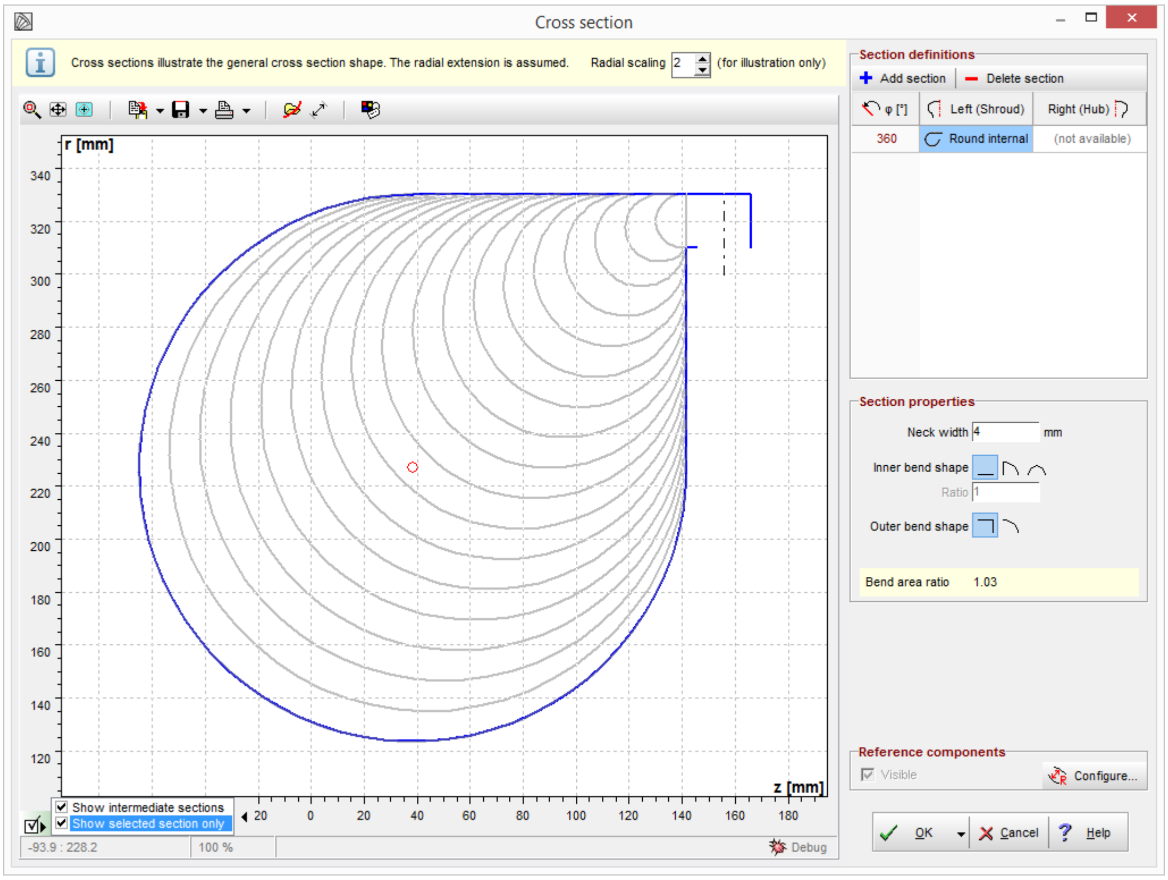


The geometry is described by the following parameters:

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	δ	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

10.2.4 Internal cross sections

Internal volutes are limited in its radial and axial extensions (see gray lines in the picture).



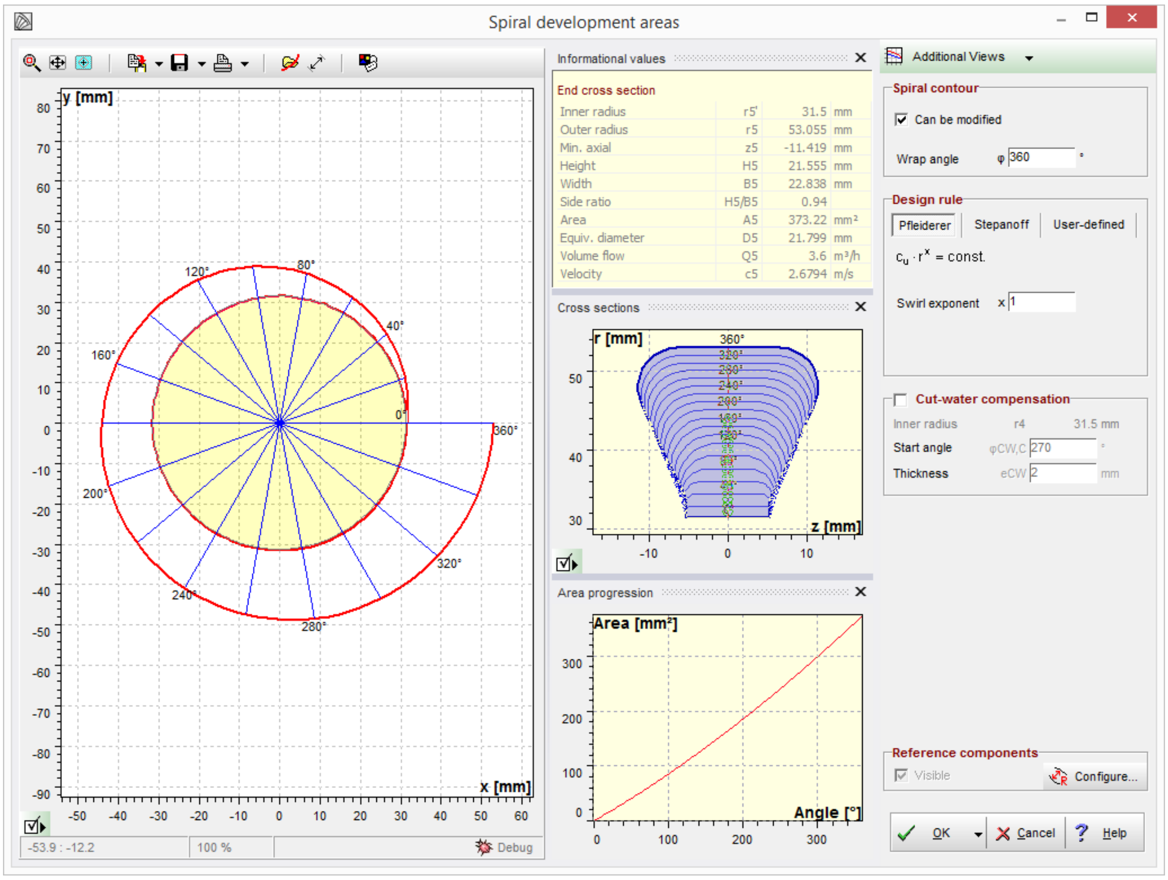
The additional bend can be described by the following parameters:

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

10.3 Spiral development areas

? Volute | Spiral development areas

The spiral development areas can be designed and calculated in this dialog box.



General

The spiral development areas can be calculated manually by pressing the **Calculate spiral** button or automatically if the **Automatic** check box is selected.

- The manual calculation freezes the radial extension of the currently designed cross sections (red contour curves in the main diagram on the left side).
  Any modifications of the Inlet definition 400 or the Cross section 406 shape result in updated cross sections while keeping the radial extension of each section constant.
  All modifications in this dialog are not considered as long as the **Calculate spiral** button is not pressed.

- The automatic mode updates the cross sections completely if anything was modified in the Inlet

definition 400 or the Cross section 406 dialog.
Furthermore all modifications in this dialog are considered directly by updating all cross sections completely.

Furthermore the **wrap angle** can be defined - default value is 360°.

Design rule

You can select the **Design rule** for volute calculation, whereas 3 possibilities exist: **Pfleiderer**, **Stepanoff**, **User-defined**.

→ Details 420 **Design Rule** 420

Cut-water compensation

In panel **Cut-water compensation** you can specify parameters for the cut-water design.

→ Details Cut-water compensation 422

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.

Note, that further calculations are based on the initial contour.

Display options

Under **Display options**, changes can be made which affect only the graphical presentation:

Show – refers to the main diagram with volute contour	
Section lines	radial angle lines
Cut-water compensation	cut-water compensation as a larger inner radius
Circle segments	circular arcs of the contour approximation

Show in cross section – refers to the cross-section diagram	
Cut-water section	cut-water cross-section
Equivalent diameter (outlet)	equivalent diameter (dashed line)
Filled cross sections	filled cross-sections

Possible warnings

Problem	Possible solutions
It's not possible to calculate spiral contour exactly. Please check "Volute/ Inlet definition" and geometry.	
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.
Volute end cross section is not reasonable. Check "Volute/ Inlet definition" and geometry.	
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.".
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 α. 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 φ > spiral wrap angle φ	Adapt circumferential position of the cross section definition ("Volute/ Cross section") or

Problem	Possible solutions
	spiral wrap angle ("Volute/ Spiral areas").
Cross sections are updated automatically. Therefore geometry modifications are possible.	
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.
Cross sections are not updated automatically. Therefore the design could be not up-to-date.	
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.

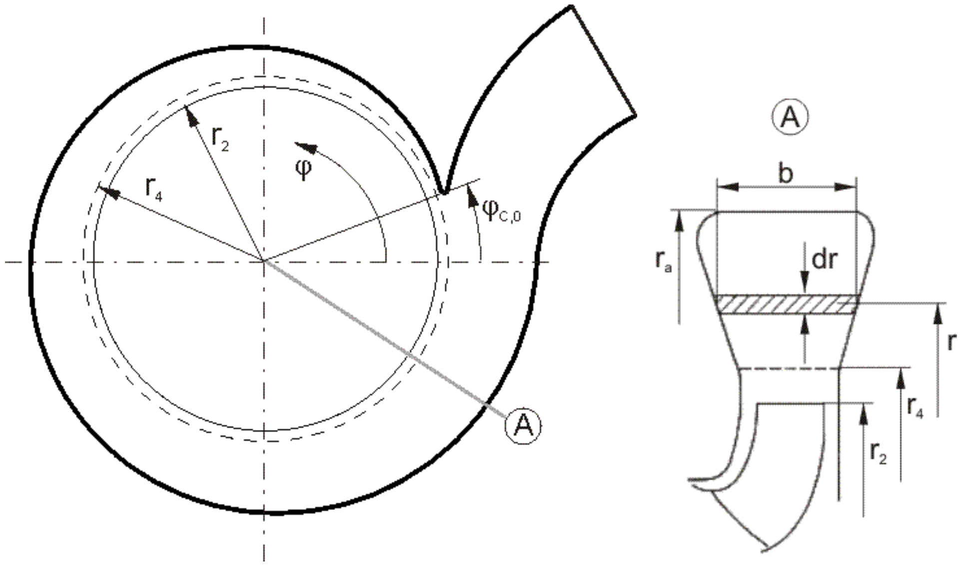
10.3.1 Design rule

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 $Q_\varphi = Q_i \cdot \varphi/(2\pi)$ 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.

1. Pfleiderer

Experience has shown that the losses can be greatly minimised 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 $c_u r^x$ = const. is obeyed. When x=1, the angular momentum is constant. For the extreme of x=0, the circular component of the absolute velocity cu remains constant at the impeller outlet.

$$\varphi = \frac{2\pi c_{u4} r_4^{\,x}}{Q_i} \int_{r_4}^{r_a(\varphi)} \frac{b(r)}{r^x} dr$$

The integral can be explicitly solved for simple cross-section shapes (rectangles, trapezoids, circles). For other, arbitrary, shapes, it can be solved numerically.

2. 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:

. The constant k_s can be determined dependent on the specific speed n_q (see Approximation function 145).

$$\varphi = \frac{2\pi k_S \sqrt{2gH}}{Q_i} \int_{r_4}^{r_a(\varphi)} b(r)\,dr$$

3. User-defined

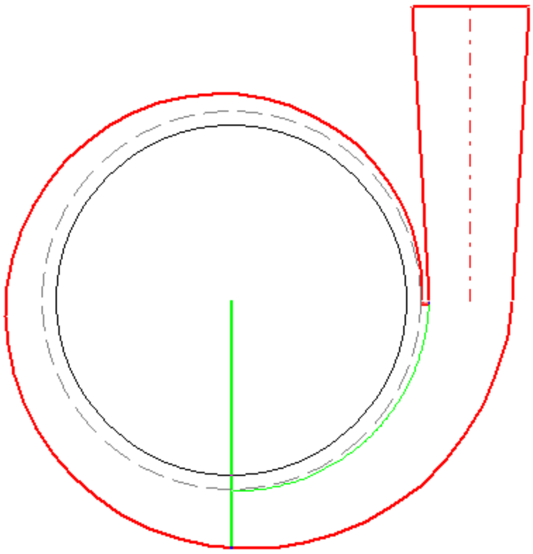
Contrary to 1. and 2. the geometry progression is defined directly. The end cross section is defined by radius or cross section area, the distribution by Radius- or Area progression (Set Progression 46).

10.3.2 Cut-water compensation

Cut-water is available for external volutes only. For internal volutes the cut-water is a result of intersection of spiral and diffuser.

Some initial cut-water parameters can be specified in the **Cut-water** section:

Inner radius r_4	Informative, see Inlet 400 r_4 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°)



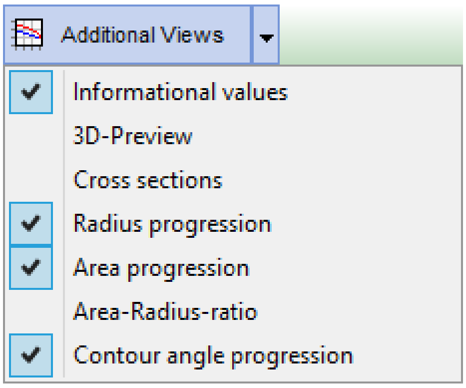
The cut-water does disturb the flow, since the cross-section of the flow is narrowed suddenly by the thickness of the cut-water.

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_4 increases linearly to a value of r_4+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.

10.3.3 Additional views

The following information can be displayed in the spiral dialog using the "Additional views" button:

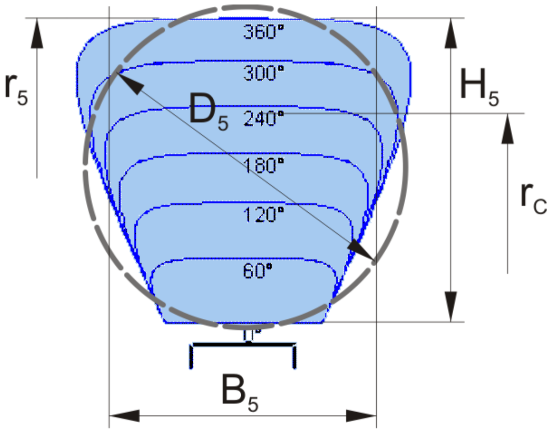


3D-Preview

[3D model](172) of the currently designed spiral development areas.

Informational values

Some informative values relating to the **end cross-section** are displayed:



Radius	r_5
Height	H_5
Width	B_5
Side ratio	H_5/B_5
Equivalent diameter	D_5
Area	A_5
Volume flow	Q_5
Average velocity	c_5
Static pressure	p_5
Density	ρ_5
Temperature	T_5
Mach-number	Ma_5

Cross sections

Volute cross sections (z-r)

Radius progression

Radius distribution (φ-r)

Area progression

Area distribution (φ-A)

Area-radius-ratio

Area/Gravity center radius (r_C) distribution (φ-A/R)

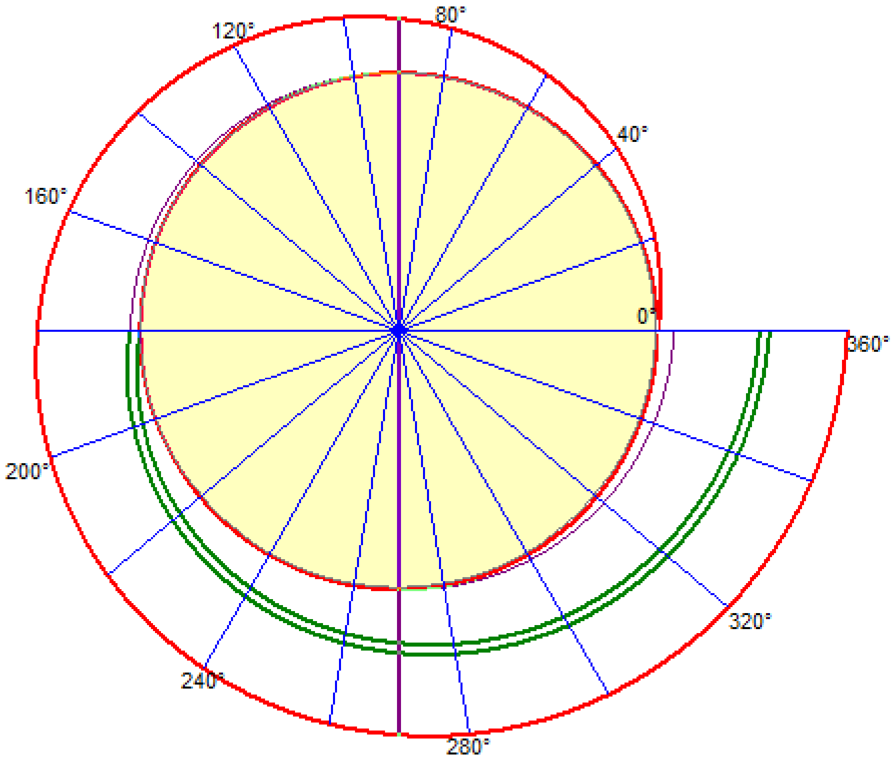
Contour angle progression

Angle between the outer spiral contour and the circumferential direction (φ-α). Note, that due to the differential characteristic of the contour angle, the continuity of this distribution is decreased by one.

10.3.4 Double Volute

Double Volutes are used to compensate asymmetric casing forces that are inevitable for Single Volutes.

Double Volute design can be activated in the initial volute Setup 401.

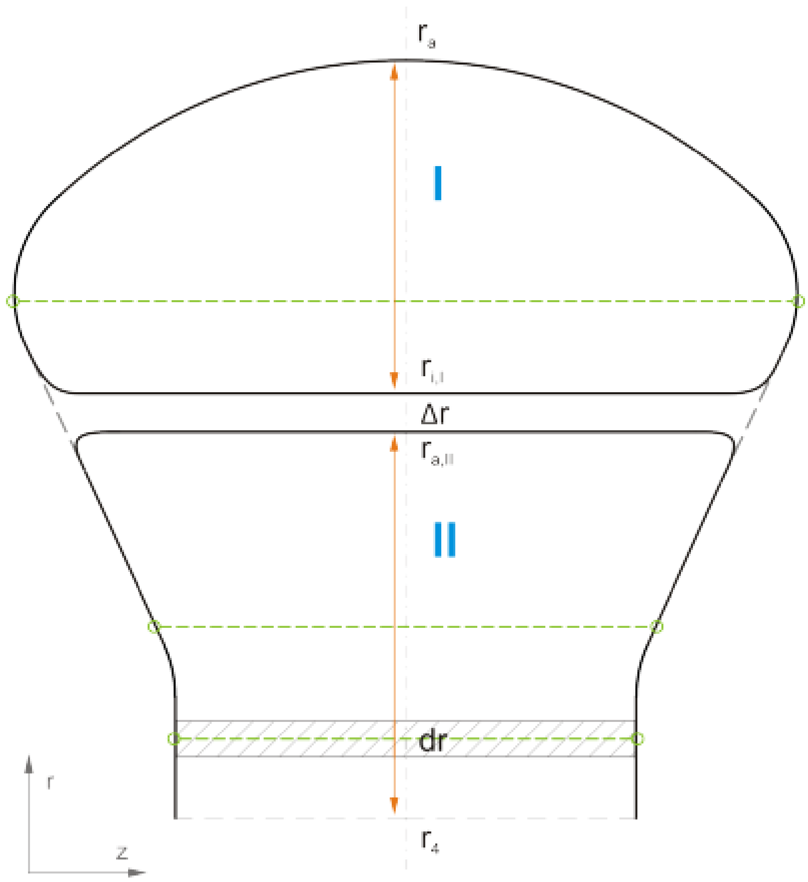


General procedure for Double Volute design

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 422. 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 $\varphi-\varphi_{Spl}$.

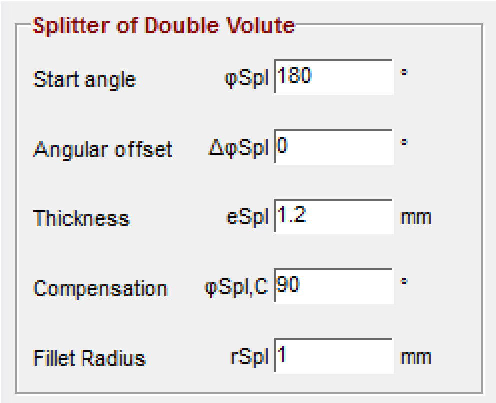
The Outer area (I) is calculated based on the Design rule 420 for
* a constant flow rate defined by the splitter start angle (normally 50% of overall flow rate)
* starting from the splitter outside radius $r_{i,I} = r_{a,II} + \Delta r$.



Splitter of Double Volute

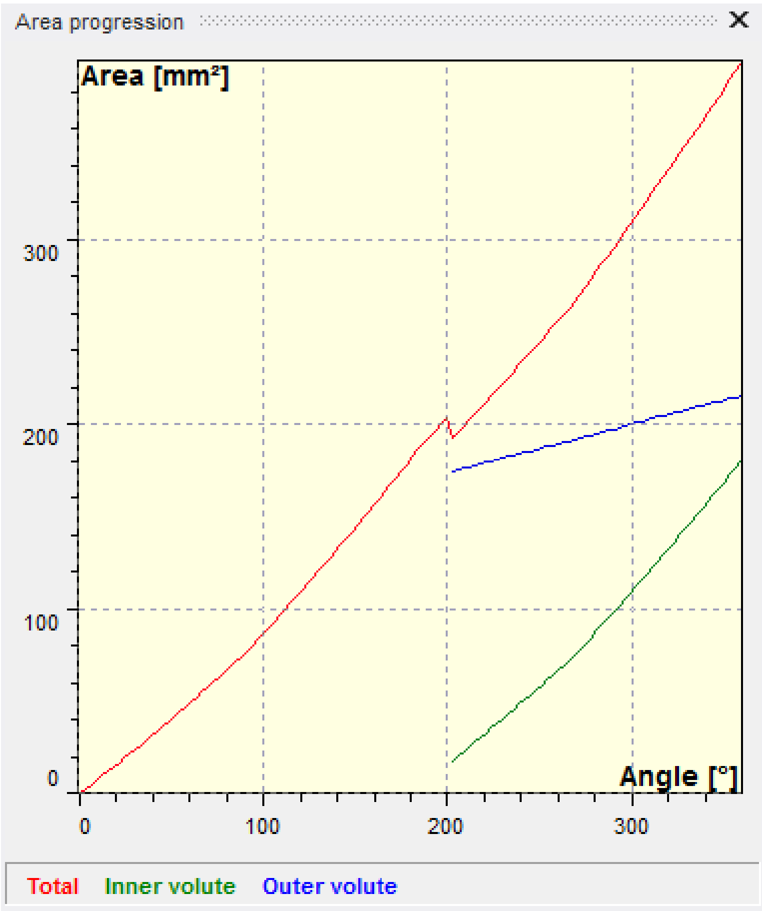
For double volutes you can define additional properties of the spiral and splitter.

- The **start angle φSpl** is the angular position where the splitter starts. It also determines the splitter contour.
- The **angular offset ΔφSpl** can be used to achieve a radial offset without changing the contour.
- The **thickness eSpl** defines the distance between the inner and outer splitter contour.
- The **compensation φSpl,C** is used analogous to the cut-water compensation.
- The **fillet radius** defines the radial corner radius between spiral and splitter surface.



Additional views

The progression diagrams contain curves for each part of the volute, like the area progression below.



Beside the default informational values 423 separate values for inner and outer part of the volute are reported.

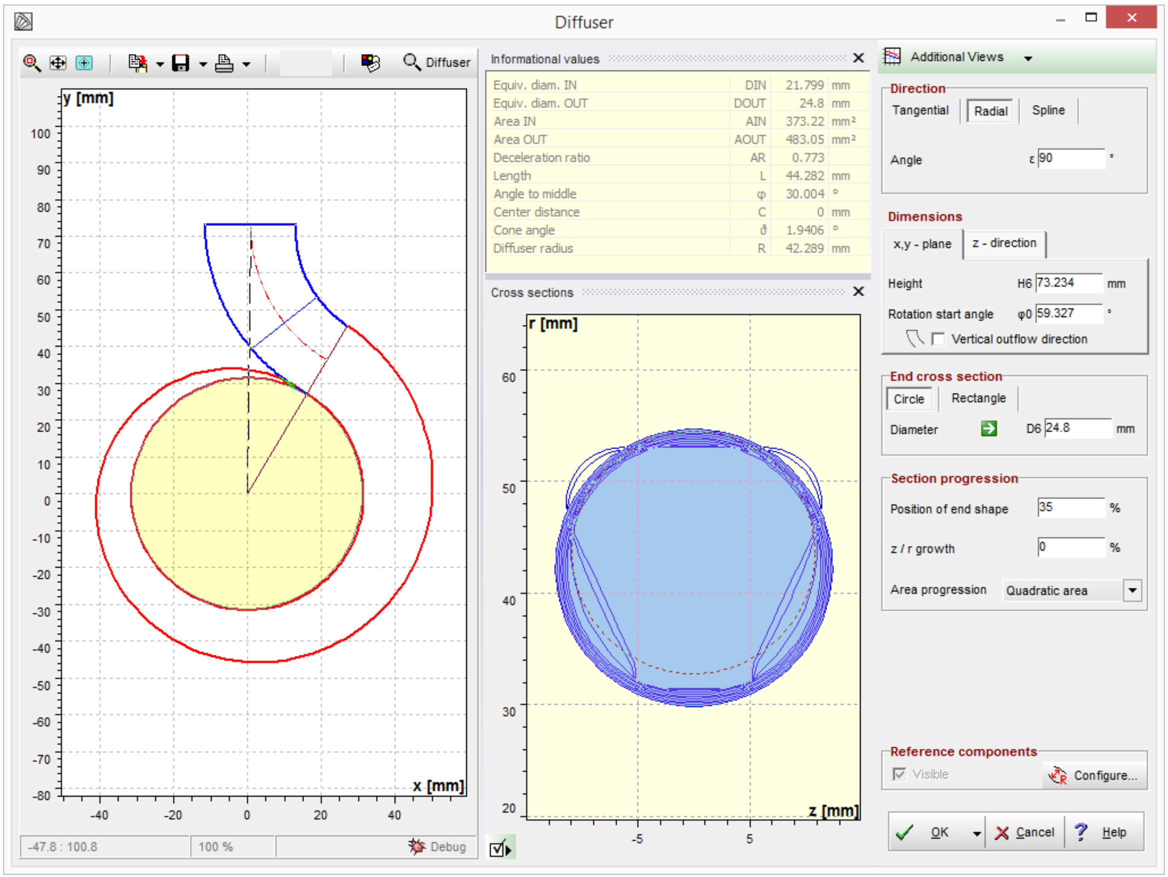
Furthermore 2 additional ratios are displayed:

- 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)

10.4 Diffuser

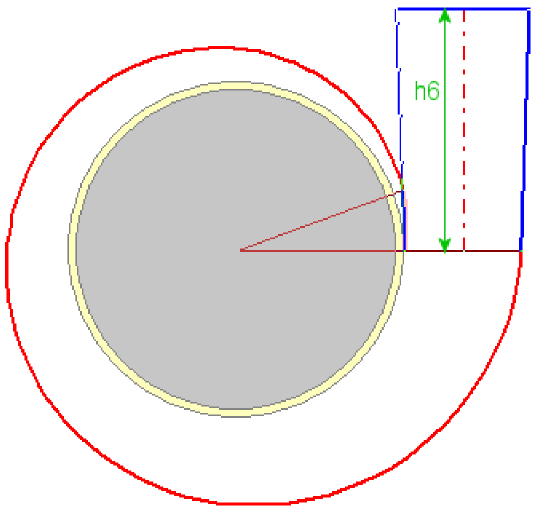
? Volute | Diffuser

The geometry of the outlet diffuser can be designed and calculated in this dialog box.

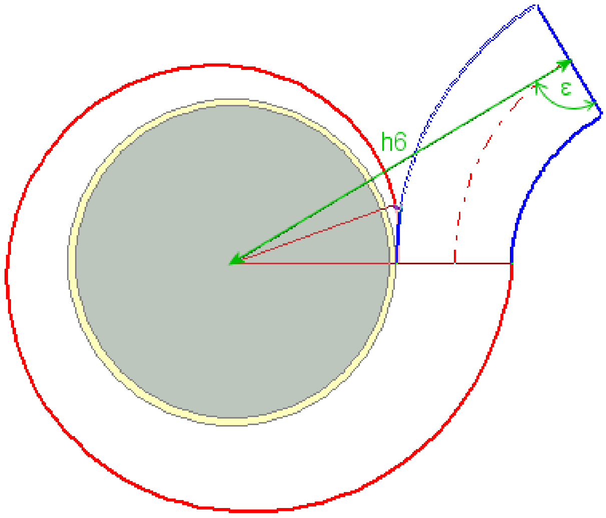


Direction

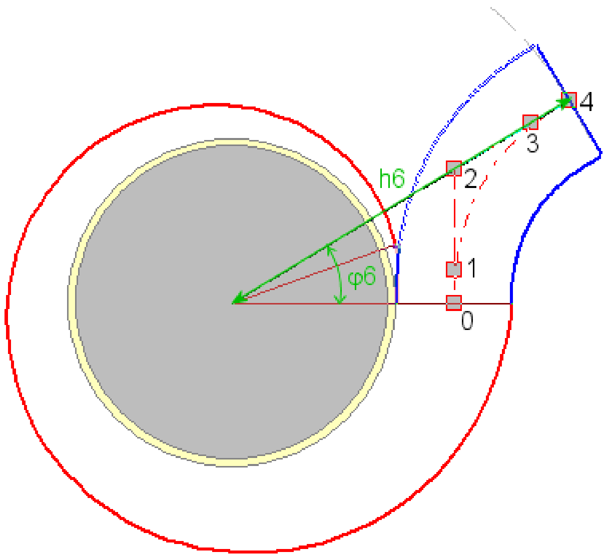
In general, 3 basic shapes are available:



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 excentricity 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 excentricity 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.

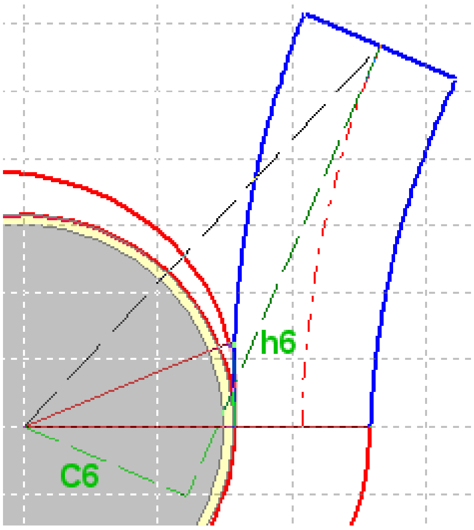
Spline diffuser

For the Spline-diffuser the angle φ_6 between connecting line impeller-center ↔ 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 $\mathbf{h_6}$, which is the distance from the diffuser outlet to a parallel line through the center point.

The distance $\mathbf{C_6}$ from the h_6-line to the center point is displayed for information, both in the diagram and numerical in the **Information** panel.

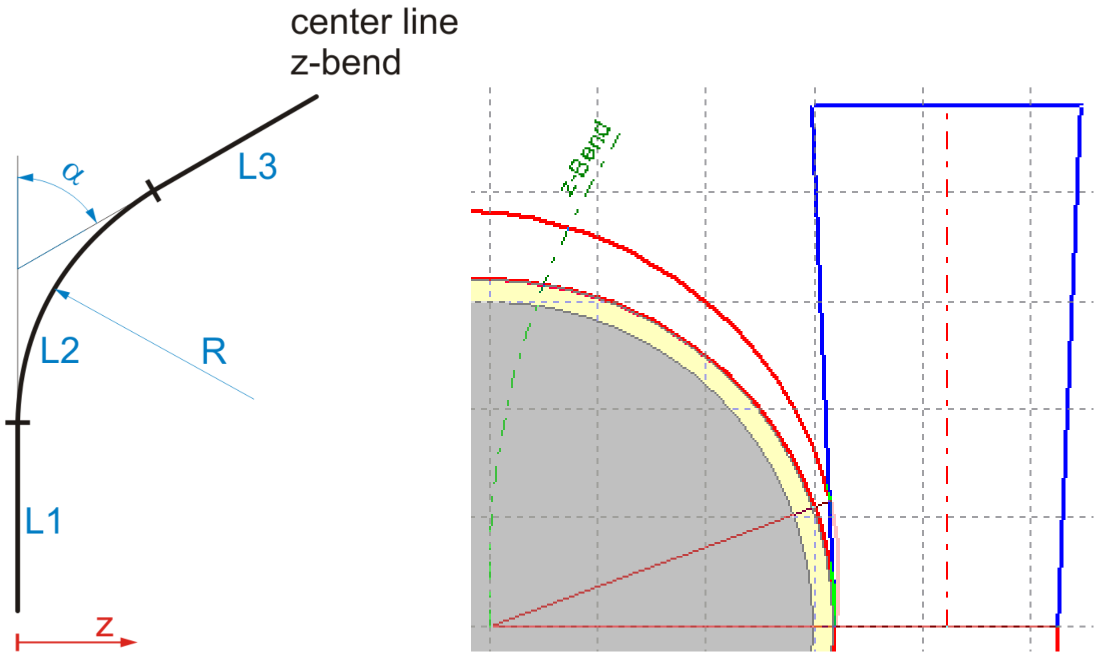
Additionally the starting position of the diffuser is defined by the angle φ_0, whereas 0° is horizontal right. The whole volute can be rotated by this value. By using the button **Vertical outflow direction** the volute can be rotated for vertical direction of the pressure joint.

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 $\mathbf{L_1}$, $\mathbf{L_2}$ and $\mathbf{L_3}$ are specified as percentage.

The curvature is defined by the radius **R**, the direction by the angle **α**.

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 $\mathbf{D_6}$ can be directly defined or selected from standard tables. In the case of a rectangular end cross-section the height $\mathbf{H_6}$ and width $\mathbf{B_6}$ 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

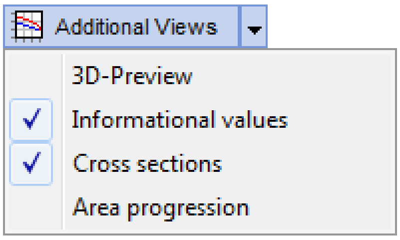
The **position of splitter end** defined the relative length of the splitter inside the diffuser.

Display options

Under **Display options**, changes can be made which affect only the graphics.

10.4.1 Additional views

The following information can be displayed in the diffuserl dialog using the "Additional views" button:



3D-Preview

3D model 172 of the currently designed diffuser geometry.

Informational values

Some informative values are displayed:

Equivalent diameter DIN	Diameter of the equivalent circle at the diffuser inlet
Equivalent diameter DOUT	Diameter of the equivalent circle at the diffuser outlet
Area AIN	Area at diffuser inlet

Area AOUT	Area at diffuser outlet
Deceleration ratio AR	$A_R = D_5^2 / D_6^2$
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_6-line to the center point
Cone angle ϑ	Cone angle from D_5 to D_6 over the length L
Diffusor radius R	Radius of middle line (for radial diffuser only)

Cross section

Volute cross sections (z-r)

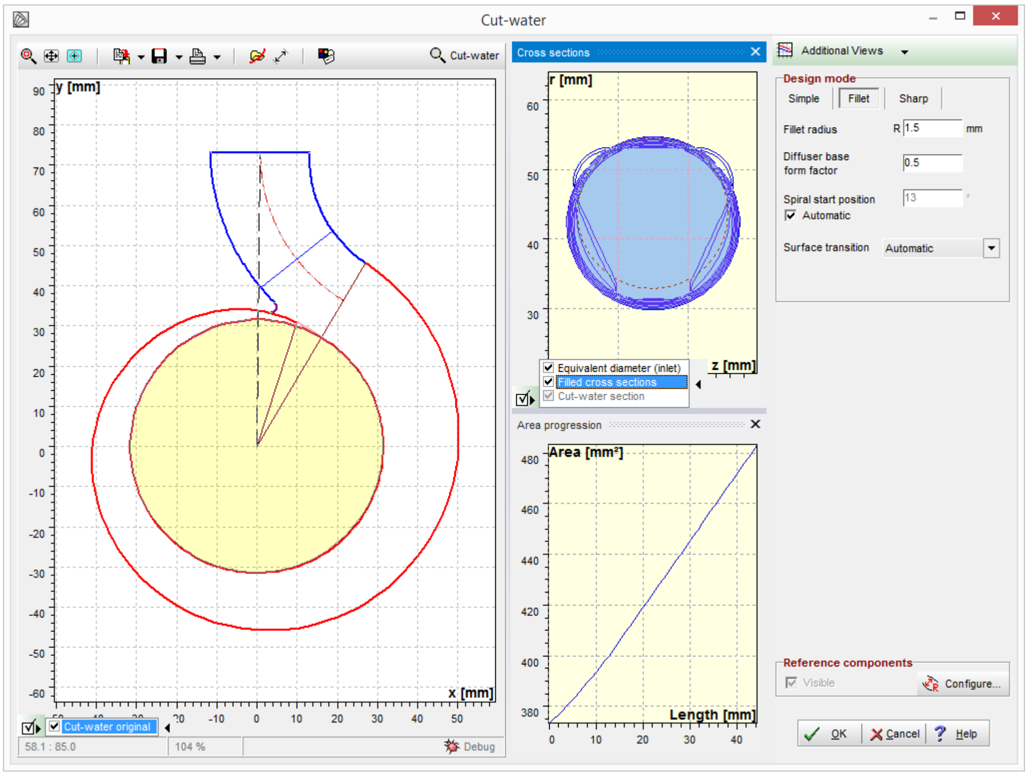
Area progression

Area distribution (l-A)

10.5 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](437)**, **[Fillet](440)** or **[Sharp](443)**.

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.

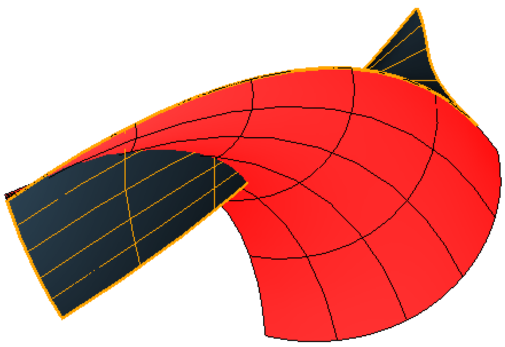
Limitations

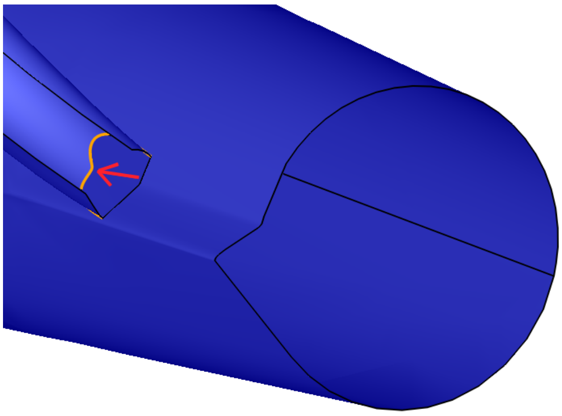
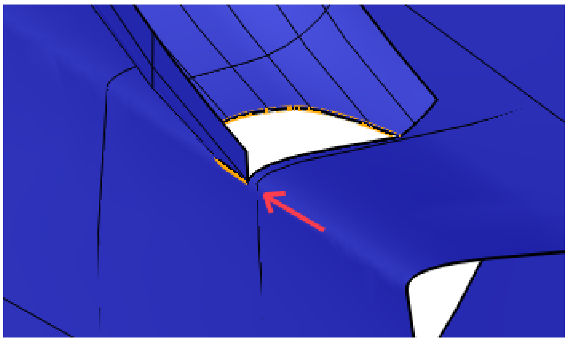
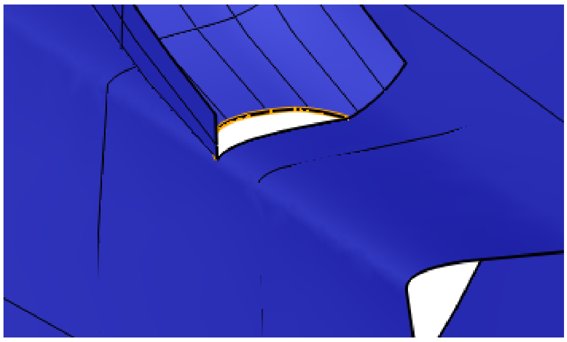
General	The wrap angle 417 must be at least 330°.
Simple	For cornered spiral cross sections the **side position** is fixed to the corner position and cannot be modified individually.
	Rounding of cut-water edges (**Round edges**) is possible only if **side position** is higher than the position of maximum 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**.

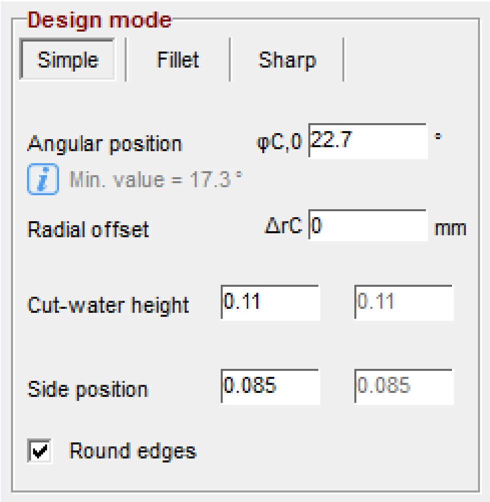
Possible warnings

Problem	Possible solutions
Cutwater is self-intersecting.	
Cut-water faces intersect each other. 	The problem might have various reasons. Therefore, modify spiral, diffuser or cutwater design. E.g. define a flat radius progression at the start of spiral development areas 417, or change **angular position** / **radial offset** of the cutwater.
3D-Error: Could not create bounded surface for Cut-water.Patch!	
Parameter **side position** is disadvantageous.	The **side position** should not be too low when edges are rounded.
3D-Error: Could not create fillet for Cut-water! Possibly, the fillet radius is too large.	

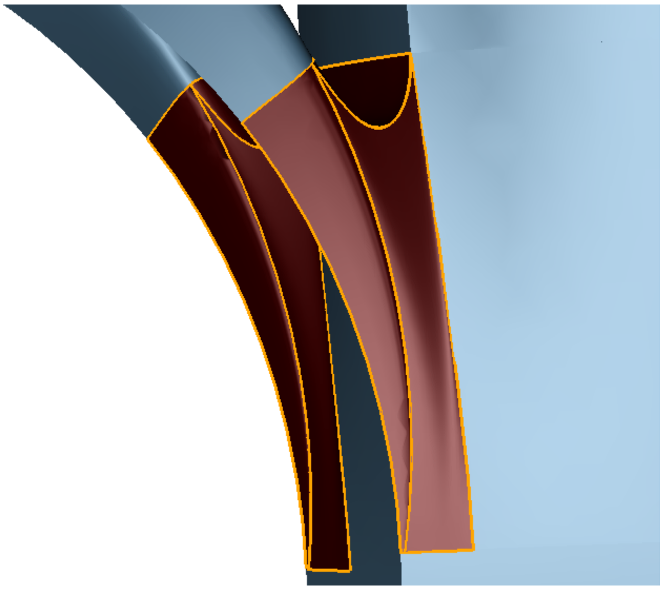
Problem	Possible solutions
[for asymmetric volutes] Fillet cannot be created because intersection curve of spiral and diffuser is wavy. 	Modify the Position of end shape 432 in the Diffuser dialog to avoid wavy intersection curve.
[for asymmetric volutes] Fillet cannot be created because intersection curve of spiral and diffuser is tangential to the sharp diffuser edge. 	Modify **Spiral start position** 

10.5.1 Simple

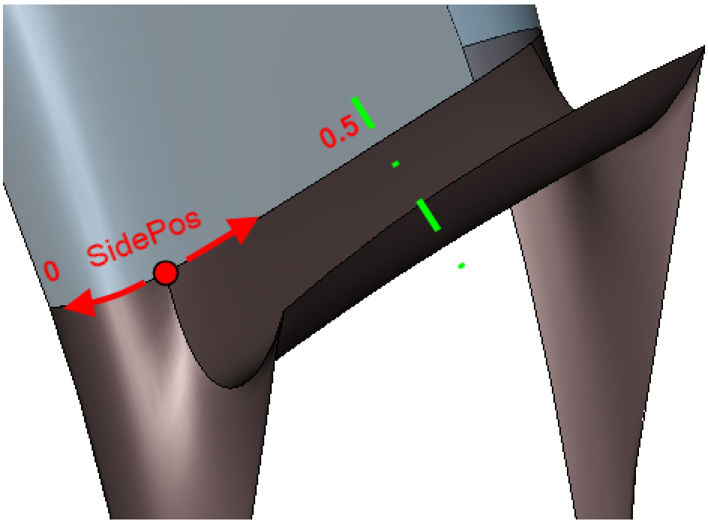
The simple cut-water is a rounding-off between spiral and diffuser.



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.

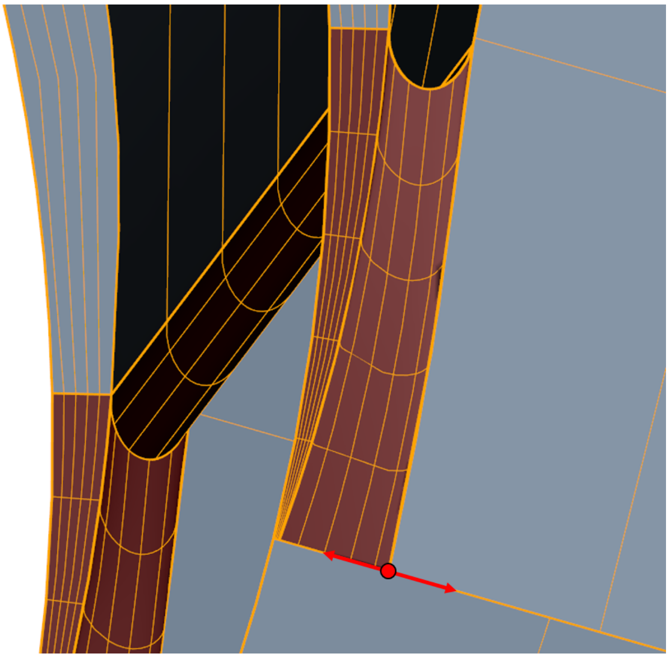


Additionally, the diffuser can be shifted in radial direction by the **radial offset** Δr_C to reduce the intersection of spiral and diffuser. This radial offset corresponds to the cut-water thickness.

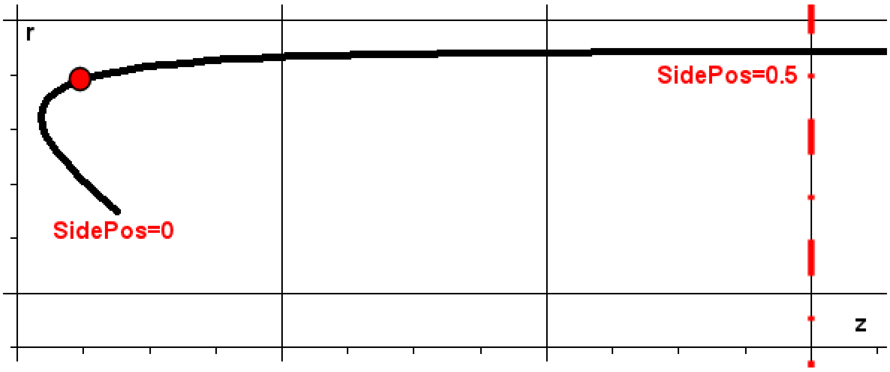


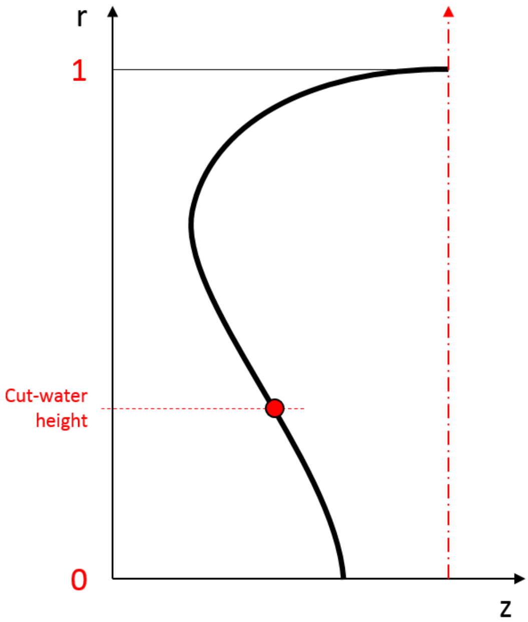
Side position defines the transition position from the central rounding surface to the side surfaces. For asymmetric spiral cross sections two independent values can be specified for left and right side.

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.

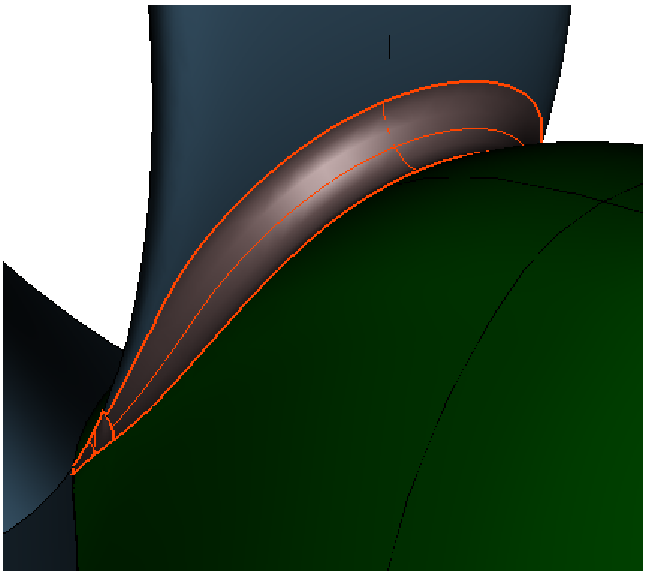




The cut-water itself is designed by a 4th order Bezier curve. The shape can be modified interactively after zooming in (**Zoom Cut-water**).

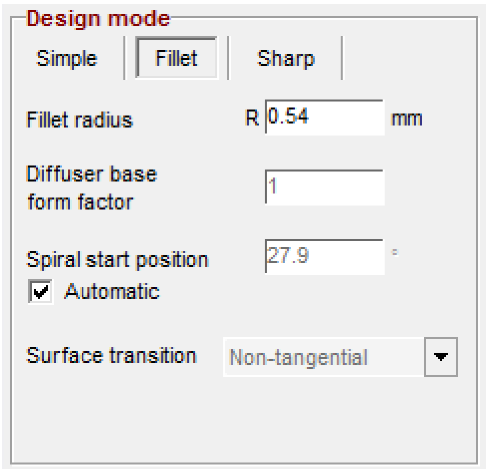
10.5.2 Fillet

For fillet cut-water design the spiral and the diffuser are trimmed and rounded at their intersection curve.



Prerequisites:

- The [wrap angle](417) must be high enough so that spiral and diffuser intersect.

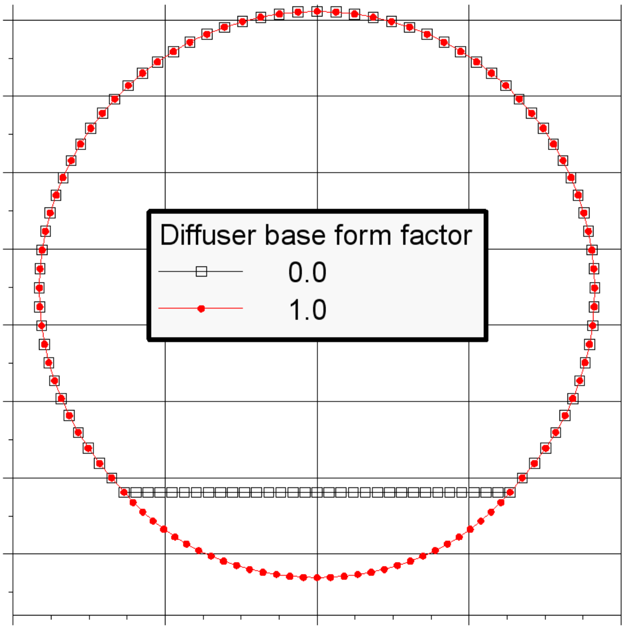


The corresponding **fillet radius** can be specified.

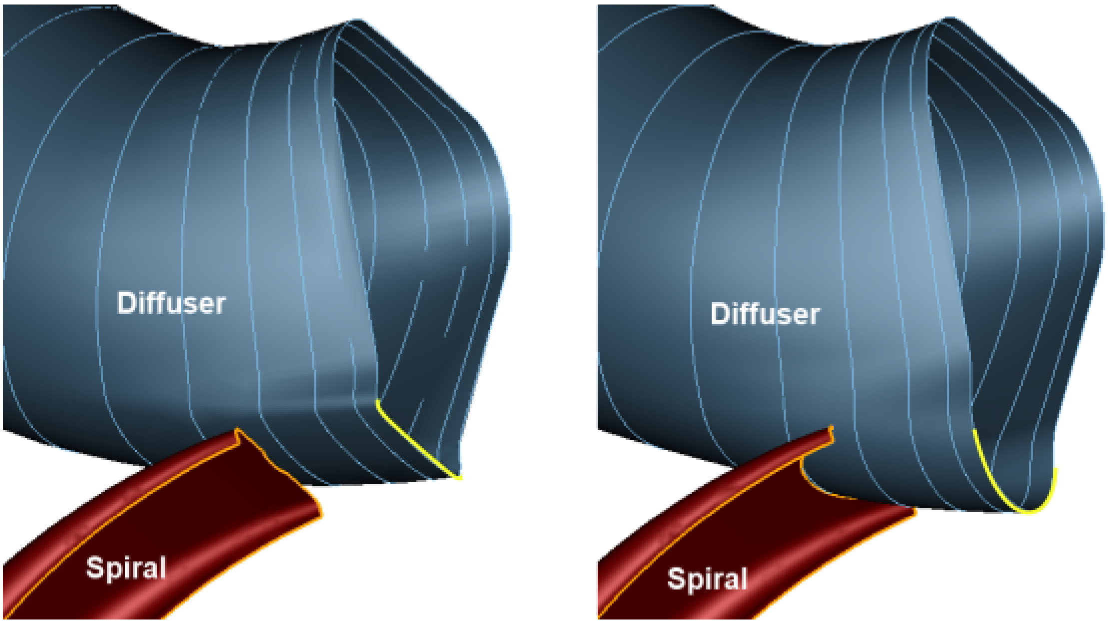
Additionally, the **Diffuser base form factor** defines the roundness of the first diffuser cross section on its base side and is between 0.2 and 1:

- 0 = cornered base side (like spiral section)
- 1 = full rounded base side

The factor affects the shape of the intersection curve and therefore the shape of the cut-water.



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

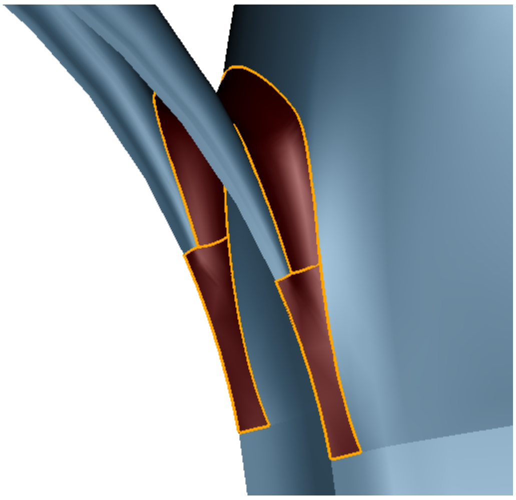
The **Spiral start position** indicates the angular position at which the spiral begins and influences the intersection of spiral and diffuser. It has to be at least 1° and must be lower than the intersection position of spiral and diffuser. If **Automatic** is activated the optimal angular position 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)

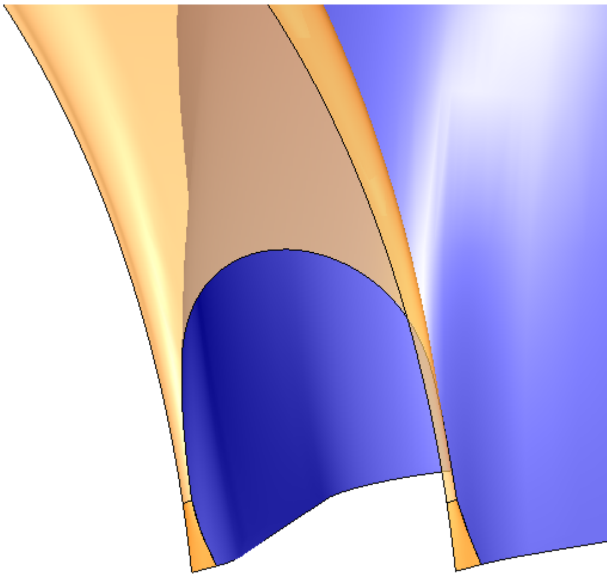
- Non-tangential: No tangential transition between both surfaces
- Automatic: Tries tangential transition. If it fails, a non-tangential transition is used. (Time-consuming)

If the fillet cut-water mode has been chosen, the **3D-model** is set to the model state 182 "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.



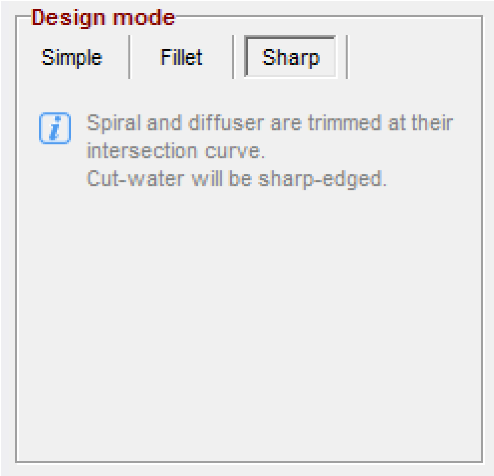
10.5.3 Sharp

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 417 must be high enough so that spiral and diffuser intersect.



10.6 CFD Setup

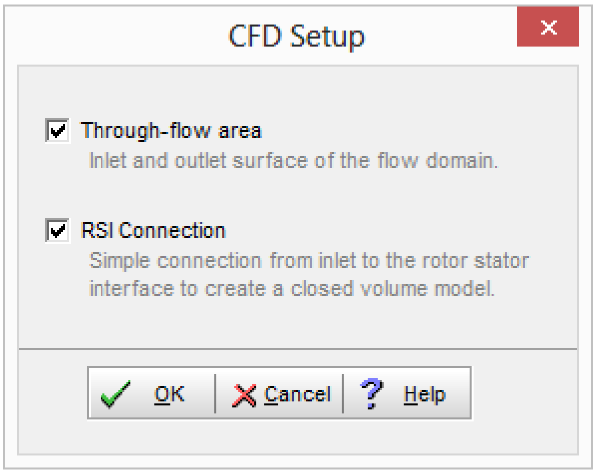
? Volute | Additional | CFD Setup

The designed geometry can be extended by **virtual** elements.

- **Through-flow area**
  Inlet and outlet surface of the flow domain.

- **RSI Connection**
  If a Rotor-Stator-Interface (RSI) is existing on the inlet side of the component, an existing gap between this RSI and the volute inlet can be closed automatically by the RSI connection.
  These surfaces provide a simplified, closed volume model for flow simulation neglecting impeller side chambers or other casing parts.

These extensions are to be used for flow simulation (CFD) and are virtual only.



10.7 Model settings

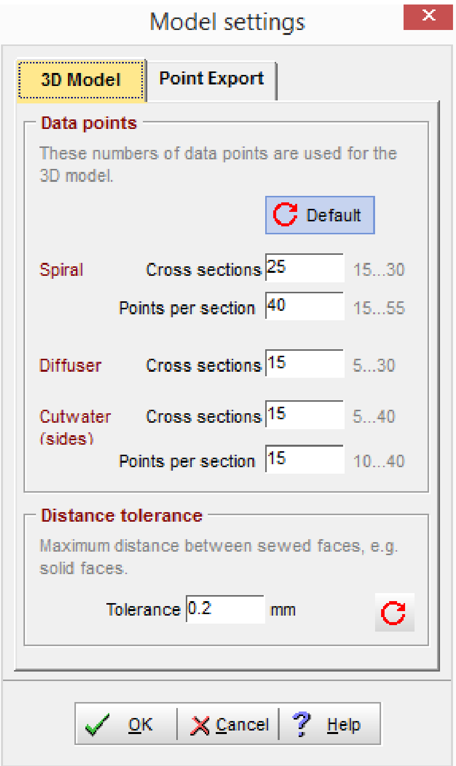
? Volute | Model settings

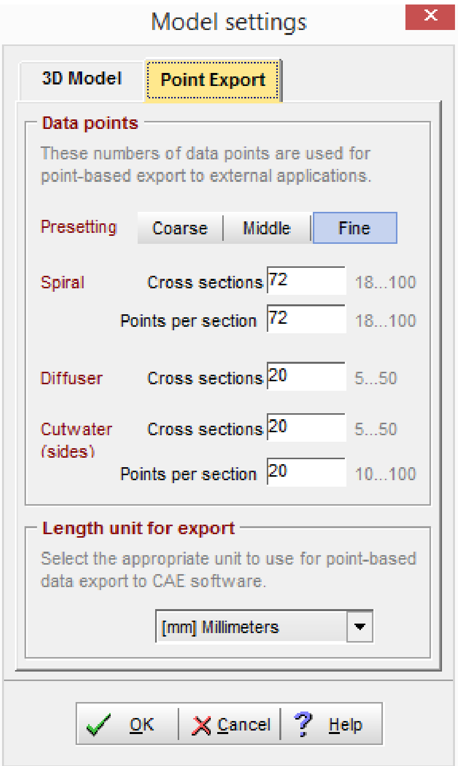
On dialog **Model settings** you can specify how many data points are to be used for the 3D model and for the point based export formats.

The number of points can be set for both cases separately for all geometry parts.

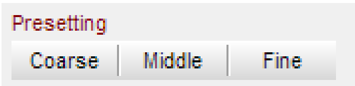
- **Spiral:** cross sections, points per cross section
- **Diffuser:** cross sections
- **Cutwater (sides):** cross sections, points per cross section

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 422*.*



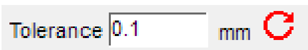


Presetting



Select from 3 global presettings.

Distance tolerance
(3D Model)

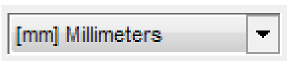


The distance tolerance defines the maximum allowed distance between sewed surfaces, e.g the faces of a solid.

If it is too small, the solids cannot be created.

If it is too big, small faces are ignored when creating a solid.

Length unit for Export
(Point Export)



The length unit for the geometry export can be selected. Please select the appropriate units when importing data to the chosen

CAD software.

When a **new volute** is created the model settings of the last opened volute are adopted.

Part XI

11 Appendix

11.1 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 Pfleiderer, 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 Bommes, Jürgen Fricke, Reinhard Grundmann
Ventilatoren
Vulkan-Verlag, 2003

Bruno Eck
Ventilatoren
Springer-Verlag, 1991

Thomas Carolus
Ventilatoren
Teubner-Verlag, 2003

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

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

Giorgio Soave

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

11.2 Symbols

Symbol	Description
α	Angle of absolute flow to u
β	Angle of relative flow to u
δ	Deviation angle flow / blade
δ_r	Swirl number
τ	Obstruction of flow channel by blades
ω	Angular velocity
ρ	Density
η	Efficiency
ψ	Pressure coefficient
σ	Thickness in circumferential direction; Speed coefficient
φ	Wrap angle; Flow coefficient
A	Cross section area
b	Width
c	Absolute velocity
c_m	Meridional velocity (c_m=w_m)
c_u	Circumferential component of absolute velocity
d	Diameter
F	Force

Symbol	Description
h	Enthalpy
H	Pump head
i	Incidence angle
L	Length
M	Torque
m	Mass flow
N	Number of revolutions
n_q, N_s	Specific speed
p	Pressure
P	Power
Q	Flow rate
r, R	Radius
s	Orthogonal thickness
S	Static moment
u	Circumferential velocity (Rotational speed)
v	Velocity
w_u	Circumferential component of relative velocity (w_u+c_u=u)
w	Relative velocity
Y	Specific energy
z	Geodetic height; Number of blades

11.3 Contact addresses

Development, Sales, Support

CFturbo Software & Engineering GmbH

www.cfturbo.com

Unterer Kreuzweg 1
01097 Dresden, Germany

Phone: (+49) 351 40 79 04 79
Fax: (+49) 351 40 79 04 80

Friedrichstraße 20
80801 Munich, Germany

Phone: (+49) 89 189 41 45 0
Fax: (+49) 89 189 41 45 20

11.4 License agreement

Software Cession and Maintenance Contract

between

CFturbo Software & Engineering GmbH

Unterer Kreuzweg 1, 01097 Dresden (Germany)

- hereinafter designated the 'Licensor' -

and

the CFturbo user

- hereinafter designated the 'User' -

§ 1 LICENSE AGREEMENT

By virtue of this agreement, the User acquires from the Licensor the non-transferable and non-exclusive right to use the software 'CFturbo' (hereinafter designated the 'Software') for a period of time, in exchange for the licence fee agreed between the Licensor and the User.

1. Licence Object

The User acquires a nodelocked license or a license for one local office network (LAN) at one distinguished location of the company.

The program package consists of a data medium (CD-ROM or DVD) with the Software and a user manual in the form of a PDF file. In the event that the Software was downloaded from the official website of the Licensor, the program package consists of the corresponding installation file including electronic documentation.

2. Duration / commencement of the licence

The User obtains the right to use the Software. The right is obtained after the payment of the full licence fee and implicitly expires at the end of the arranged time period.

4. Right of Use

(1) In accordance with this contract, the Licensor grants the User a right of use to the Software described under 1. as well as a right to use the necessary printed matter and documentation. The printing-out of the manual for the purposes of working with the Software is permitted.

(2) The User may duplicate the Software only insofar as the duplication in question is necessary for the use of the Software. Necessary reasons for duplication notably include the installation of the Software from the original data medium onto the mass storage of the hardware used, as well as the loading of the Software into the RAM memory.

(3) The User is entitled to perform duplication for backup purposes. However, in principle, only a single backup copy may be created and stored. The backup copy must be labelled as being a backup copy of the ceded Software.

(4) If, for reasons of data security or the assurance of a fast reactivation of the computer system after a total failure, the regular backing-up of the entire dataset including the computer programs used is essential, then the User may create the number of backup copies which are compulsorily required. The data media concerned must be labelled accordingly. The backup copies may only be used for purely archival purposes.

(5) The User is obliged to take appropriate measures to prevent the unauthorized access of third parties to the program including its documentation. The supplied original data media, as well as the backup copies, must be stored in a location protected against the unauthorized access of third parties. The employees of the User must be explicitly encouraged to observe these contractual conditions as well as the provisions of copyright law.

(6) Additional duplications, also including the printing-out of the program code on a printer, must not be created by the User. The copying and the handover or transfer of the user manual to third parties

is not permitted.

5. Multiple Use and Networks

(1) The User may use the Software on any hardware available to him, provided that this hardware is appropriate for the use according to the Software documentation. In the event of changing the hardware, the Software must be erased from the previously used hardware.

(2) The simultaneous reading in, storage or use on more than one hardware device is not permitted unless the User has acquired multiple-use licences or network licences. Should the User wish to use the Software on multiple hardware configurations at the same time, for example to permit the use of the Software by several employees, he must purchase the corresponding number of licences.

(3) The use of the ceded Software on different computers on a network or another multiple-workstation computer system is permitted, provided that the User has purchased multiple-use licences or network licences. If this is not the case, the User may only use the Software on a network if he prevents simultaneous multiple use by means of access protection mechanisms.

6. Program Modifications

(1) The disassembly of the ceded program code into other code forms (decompilation) as well as other types of reverse-engineering of the different manufacturing stages of the software, including a modification of the program, is not permitted.

(2) The removal of the copy protection or similar protection mechanisms is not permitted. Insofar as the trouble-free use of the program is impaired or hindered by one of the protection mechanisms, the Licensor is obliged to remedy the fault on an appropriate request. The User bears the burden of proof of the impairment or hindrance of trouble-free usability as a result of the protection mechanism.

(3) Copyright notices, serial numbers and other marks used for program identification purposes must in no event be removed or modified. This also applies to the suppression of the screen display of such marks.

7. Resale and Leasing

Resale and leasing of the Software or other cession of the Software to third parties is only permitted with the written agreement of the Licensor.

8. Warranty

(1) The Licensor makes no warranty with respect to the performance of the Software or the obtained data and the like. He grants no guarantees, assurances or other provisions and conditions with respect to the merchantability, freedom from defects of title, integration or usability for specific purposes, unless they are legally prescribed and cannot be restricted.

(2) Defects in the ceded software including the user manuals and other documents must be remedied by the Licensor within an appropriate period of time following the corresponding notification of the defect by the User. The defect is remedied by free-of-charge improvements or a replacement delivery, at the discretion of the Licensor.

(3) For the purposes of testing for and remedying defects, the User permits the Licensor to access the Software via telecommunications. The connections necessary for this are established by the User according to the instructions of the Licensor.

(4) A right of cancellation of the User due to the non-granting of use according to § 543 para. 2 clause 1 no. 1 of the Civil Code is excluded insofar as the improvement or replacement delivery is not to be regarded as having failed. Failure of the improvement or replacement delivery is only to be assumed if the Licensor was given sufficient opportunity to make the improvement or replacement delivery.

(5) Furthermore, the statutory regulations also apply.

9. Liability

(1) The claims of the User for compensation or replacement of futile expenditure conform, without regard to the legal nature of the claim, to the existing clause.

(2) In the Software, it is a question of a design procedure. It is considered to be purely an approximation method. The Licensor is not liable for the functioning of the data obtained in practice, for the manufactured prototypes or components, or for possible consequential damages resulting therefrom.

(3) The Licensor is liable for damage involving injury to life and limb or to health, without limitation, insofar as this damage is the result of a negligent or intentional breach of obligation on the part of the Licensor or one of his legal representatives or vicarious agents.

(4) Otherwise, the Licensor is liable only for gross negligence and deliberate malfeasance.

(5) Liability for consequential damages due to defects is excluded.

(6) The above regulations also apply in favour of the employees of the Licensor.

(7) The liability according to the Product Liability Act (§ 14 ProdHaftG) remains unaffected.

(8) The liability of the Licensor regardless of negligence or fault for defects already existing on entering into the contract according to § 536 a para. 1 of the Civil Code is expressly excluded.

10. Inspection Obligation and Notification Obligation

(1) The User will inspect the delivered Software including its documentation within 8 working days after delivery, in particular with regard to the completeness of the data media and user manuals as well as the functionality of the basic program functions. Defects determined or detectable hereby must be reported to the Licensor within a further 8 working days by means of a registered letter. The

defect notification must contain a detailed description of the defects.

(2) Defects which cannot be detected in the context of the described appropriate inspection must be reported within 8 working days of their discovery with observance of the notification requirements specified in paragraph 1.

(3) In the event of the violation of the inspection and notification obligation, the Software is considered to be approved with regard to the defect concerned.

11. Intellectual Property, Copyright

The Software and all the authorized copies of this Software made by the User belong to the Licensor and are the intellectual property of the latter. The Software is legally protected. Insofar as it is not expressed stated in this contract, the User is granted no ownership rights to the Software, and all rights not expressly granted by means of this contract are reserved by the Licensor.

12. Return

(1) At the end of the contractual relationship, the User is obliged to return all of the original data media as well as the complete documentation, materials, and other printed matter ceded to him. The program and its documentation must be delivered to the lessor free of charge.

(2) The appropriate return also includes the complete and final deletion of all installation files and online documentation, as well as any copies that may exist.

(3) The Licensor may dispense with the return and order the deletion of the program and the destruction of the documentation. If the Licensor exercises this elective right, he will explicitly inform the User to this effect.

(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 seconds competent employees for this purpose, who are authorized to pronounce and decide on defects, functional expansions, functional cutbacks and modifications to the program structure. If necessary, other work involving the computer system must be discontinued during the time of the maintenance work.

(4) The customer grants the Licensor access to the Software via telecommunications. The connections necessary for this are established by the customer according to the instructions of the Licensor.

5. Liability

(1) The Licensor is liable only for deliberate malfeasance and gross negligence and also that of his legal representatives and managerial staff. For the fault of miscellaneous vicarious agents, the liability is limited to five times the annual maintenance fee as well as to such damage the arising of which is typically to be expected in the context of software maintenance.

(2) The liability for data loss is limited to the typical data retrieval expenditure which would have come about in the regular preparation of backup copies in accordance with the risks.

§ 3 MISCELLANEOUS AGREEMENTS

1. Conflicts with Other Terms of Business

Insofar as the User also uses General Terms of Business, the contract comes about even without express agreement about the inclusion of General Terms of Business. Insofar as the different General Terms of Business coincide with respect to their content, they are considered to be agreed. The regulations of the anticipated law replace any contradictory individual regulations. This also applies to the case in which the Conditions of Business of the User contain regulations which are not contained in the framework of these Conditions of Business. If the existing Conditions of Business contain regulations not contained in the Conditions of Business of the User, then the existing Conditions of Business apply.

2. Written Form

All agreements which contain a modification, addition or substantiation of these contractual conditions, as well as specific guarantees and stipulations, must be set down in writing. If they are declared by representatives or vicarious agents of the Licensor, they are only binding if the Licensor has granted his written consent to them.

3. Notice and Cognizance Confirmation

The User is aware of the use of the existing General Conditions of Business on the part of the Licensor. He has had the opportunity to take note of their content in a reasonable manner.

4. Election of Jurisdiction

In relation to all of the legal relations arising from this contractual relationship, the parties agree to apply the law of the Federal Republic of Germany, with the exception of the United Nations Convention on Contracts for the International Sale of Goods.

5. Place of Jurisdiction

For all disputes arising in the context of the execution of this contractual relationship, Dresden is agreed to be the place of jurisdiction.

6. Severability Clause

Should one or more of the provisions of this contract be ineffective or void, then the effectiveness of the remaining provisions remains unaffected. The parties undertake to replace the ineffective or void clauses with legally effective ones which are as equivalent as possible to the originally intended economic result. The same applies if the contract should contain a missing provision which requires addition.

Index

- 1 -

- 3 -

- A -

- B -

- C -

- D -

- E -

- F -

- S -

- T -

- U -

- V -

- W -

- Z -