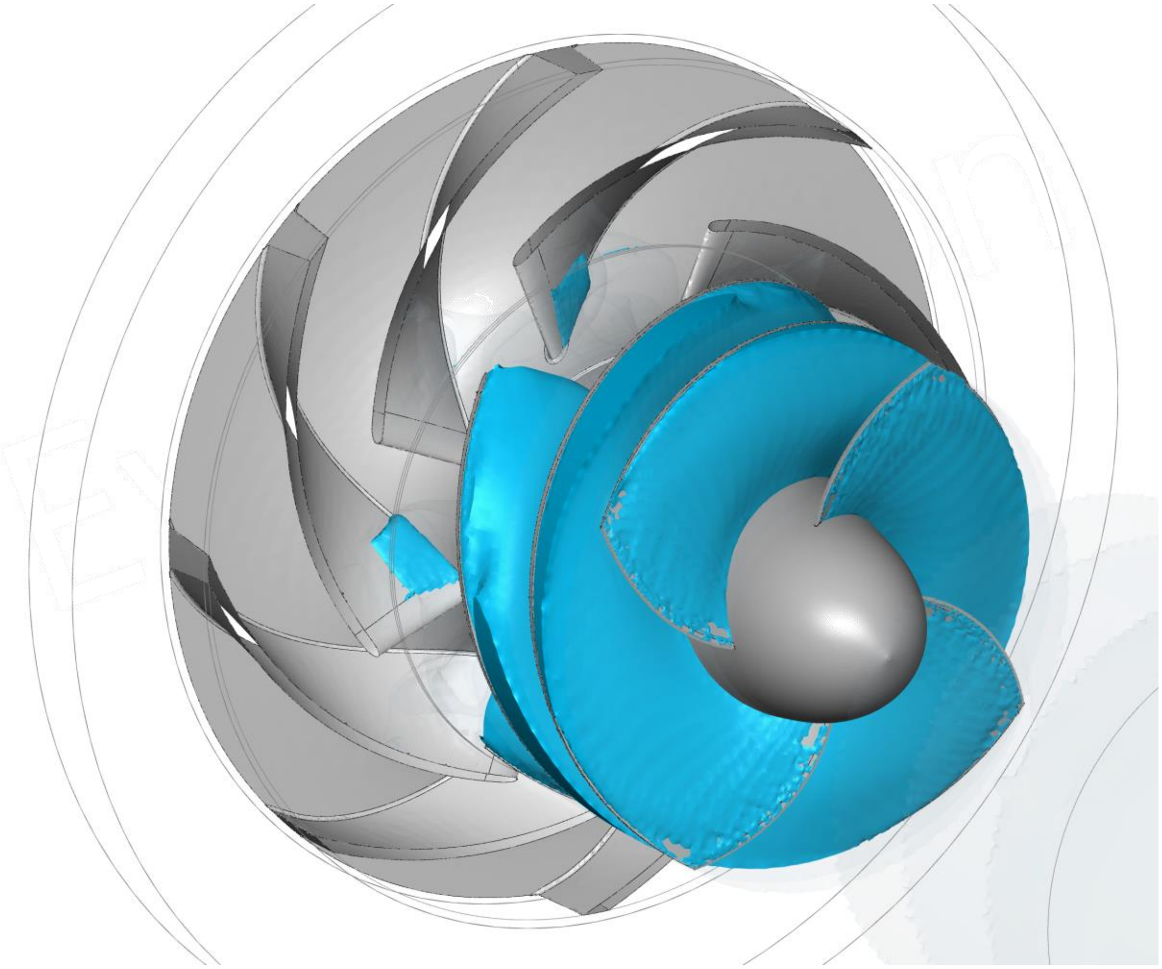
2025 Series - Case Study #1

CFturbo BLADERUNNER

Layout of an oxidizer centrifugal pump with consideration of cavitation



Cavitation bubbles on an inducer blading calculated by CFD

Powerful liquid propellant rocket engines use a turbopump to pump fuel and oxidizer from the tanks to the inlet of the combustion chamber, where a pressure level of 100 to over 300 bar is reached. The fuel used is usually liquid hydrogen (LH2), a special kerosene (rocket propellant 1, RP-1) or, more recently, liquid methane (LCH4). In most cases, liquid oxygen (LOX) is used as the oxidizer. The two liquids are pumped and pressurized by two centrifugal pumps mounted on a common shaft and driven by a gas turbine forming the turbopump. For better adaptation to very different mass flows, two separate turbopumps are sometimes also installed, such as in the new SpaceX Raptor engine.

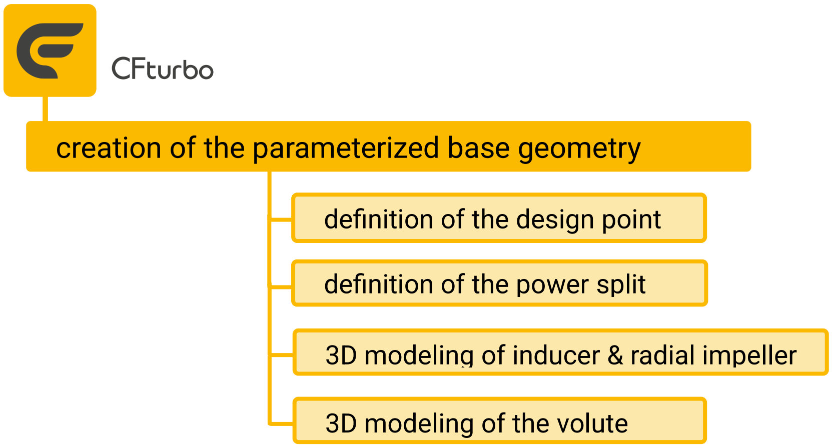
For weight reasons, turbopumps have a very compact design despite large mass flows, so that the hydraulic power must be applied via a very high rotational speed of the centrifugal pumps. As at least one fluid is cryogenic, i.e. at a very low temperature, they are also exposed to extreme thermal loads.

The study shows the design and optimization of a LOX centrifugal pump with CFturbo 2025 and CFturbo BLADERUNNER 2025 for a fictional rocket engine powered by LOX/RP-1 (Kerolox), which delivers around 250 kN thrust at an exit velocity of 3,000 m/s. Current designs such as the medium-weight SpaceX Falcon 9 launcher or the much smaller RFA ONE from Rocket Factory Augsburg, which is still under development, use nine engines in the first rocket stage. Nine of the fictional engines would be enough to launch a payload of 3,400 kg into low earth orbit (LEO).

The turbopump is the heart of a rocket engine. It must constantly supply the combustion chamber with fuel and oxidizer with absolute reliability while maintaining the high combustion chamber pressure. When designing the LOX centrifugal pump, attention must therefore be paid to a phenomenon whose occurrence cannot be completely avoided, but can certainly be controlled: Cavitation, i.e. the formation of vapor bubbles when the pressure falls below the vapor pressure.

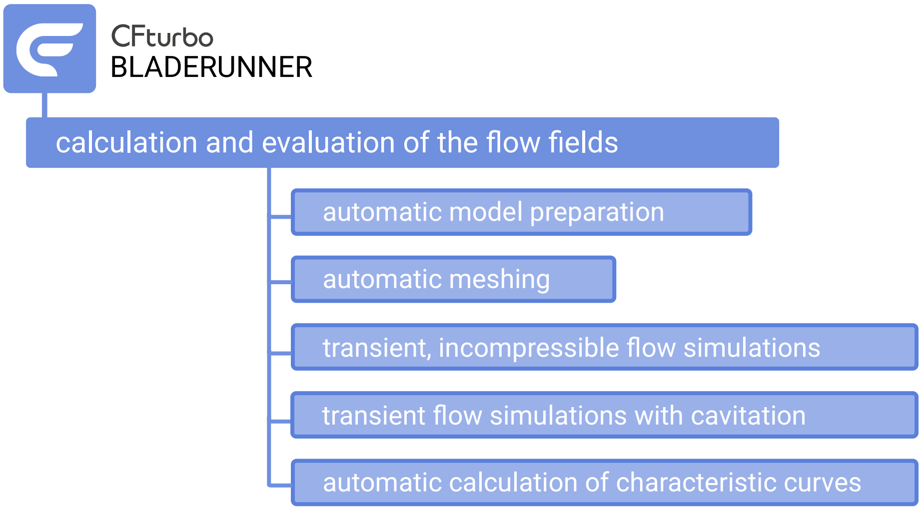
Cavitation often occurs at the leading edges of the blades of radial impellers, where local acceleration of the fluid creates so-called suction peaks. In the centrifugal pumps of a turbopump, it is therefore common practice to install an axial impeller, known as an inducer, upstream of a radial impeller, which already builds up an initial pressure and thus restricts cavitation locally.

CFturbo 2025 enables the design and 3D drafting of the entire stage, consisting of an inducer, a radial impeller and a surrounding volute, in guided individual steps:



Immediately after the 3D modelling in CFturbo 2025, CFturbo BLADERUNNER 2025 performs a series of flow simulations (CFD) for various operating points and, after a short time, displays characteristic curves that precisely characterize the behavior of the stage.

New to CFturbo BLADERUNNER 2025 are transient flow simulations in which the rotating components are moved from one calculation step to the other. This increases the accuracy of the results, and highly fluctuating processes such as cavitation are captured realistically.



CFturbo BLADERUNNER

- calculation and evaluation of the flow fields
  - automatic model preparation
  - automatic meshing
  - transient, incompressible flow simulations
  - transient flow simulations with cavitation
  - automatic calculation of characteristic curves

calc

Determination of the design data

Calculation of the LOX mass flow, determination of the pressure increase

The total mass flow of fuel and oxidizer required to generate thrust depends, among other things, on the reaction enthalpy of the fuel, the reaction speed achieved in the combustion chamber and the exit speed at the thrust nozzle. It can be calculated using the specific impulse I_s, which defines the ratio of the generated thrust F to the required mass flow $\dot{m}_{tot}$ and is historically specified in seconds in rocket technology [1]. For the combination LOX/RP-1, a value of approximately $I_{s,\,LOX/RP\text{-}1}$ = 275 s is achieved at sea level.

$$I_{S,LOX/RP-1} = \frac{F}{\dot{m}_{tot}} = 275 \cdot 9{,}81 \frac{Ns}{kg} = 2698 \frac{Ns}{kg} \overset{\text{def}}{=} 275\ s$$

With an oxidizer-to-fuel ratio of 2.56, the LOX mass flow rate results in

$$\frac{\dot{m}_{LOX}}{\dot{m}_{RP-1}} = 2{,}56 \quad , \quad \dot{m}_{tot} = \dot{m}_{LOX} + \dot{m}_{RP-1} \quad , \quad \dot{m}_{LOX} = \frac{2{,}56}{2{,}56+1} \dot{m}_{tot} = 0{,}719\, \dot{m}_{tot}$$

$$\dot{m}_{LOX} = 0{,}719 \frac{\mathrm{F}}{\mathrm{I_{S,LOX/RP-1}}} = 0{,}719 \cdot \frac{250\ kN}{2698\ Ns/kg} = 66{,}6 \frac{kg}{s}$$

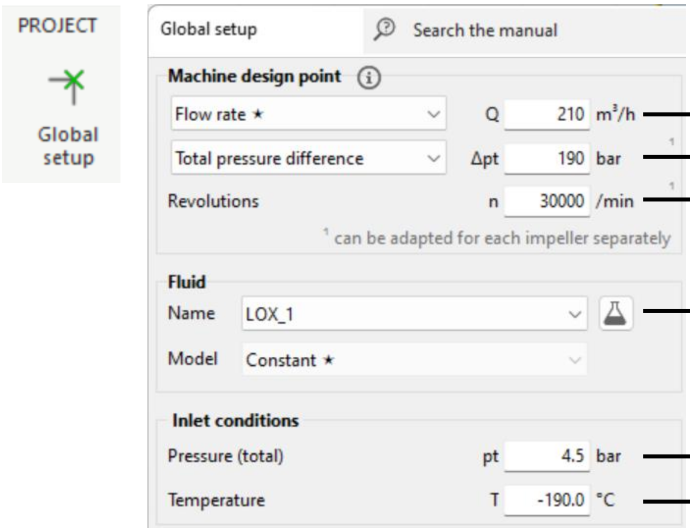
$$\dot{V}_{LOX} = \frac{\dot{m}_{LOX}}{\rho_{LOX}} = \frac{66{,}6\ kg/s}{1141{,}9\ kg/m^3} \cdot 3600 \frac{s}{h} = 210 \frac{m^3}{h}$$

The combustion chamber pressure should reach 150 bar. As the liquid oxygen enters the combustion chamber at overpressure, but first cools the combustion chamber wall and the thrust nozzle and then drives the gas turbine of the turbopump, a required pressure increase of $\Delta p_{t,\,LOX}$ = 190 bar is assumed for the LOX centrifugal pump.

Creation of the Parameterized Base Geometry within CFturbo 2025
3D Modeling of Inducer and Radial Impeller

The speed of the inducer and radial impeller is set to a standard value of n = 30,000 rpm. As the tanks for fuel and oxidizer are designed with thin walls to save weight, the static pressure of the fluids is limited to 4 to 5 bar. The total pressure at the inducer inlet is therefore estimated at $\Delta p_{t,\,LOX}$ = 4.5 bar (50 psig) for operation on the ground without the acceleration forces after the launch of the rocket.

This design data is entered in the global setup of the project:



Machine design point

Flow rate ★ Q 210 m³/h

Total pressure difference Δpt 190 bar

Revolutions n 30000 /min

[1] can be adapted for each impeller separately

Fluid

Name LOX_1

Model Constant ★

Inlet conditions

Pressure (total) pt 4.5 bar

Temperature T -190.0 °C

LOX flow rate

total pressure difference $\Delta p_{t,\,LOX}$

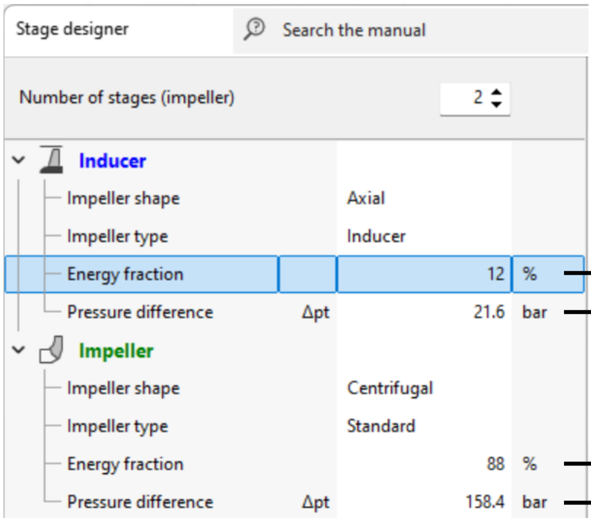
rotational speed

substance data from CoolProp stored in CFturbo

inlet total pressure

LOX temperature

The power split between the inducer – typically 8% to 12% – and the radial impeller and thus the pressure build-up in the two components is then defined:



Number of stages (impeller) 2

Inducer

Impeller shape		Axial	
Impeller type		Inducer	
Energy fraction		12	%
Pressure difference	Δpt	21.6	bar

Impeller

Impeller shape		Centrifugal	
Impeller type		Standard	
Energy fraction		88	%
Pressure difference	Δpt	158.4	bar

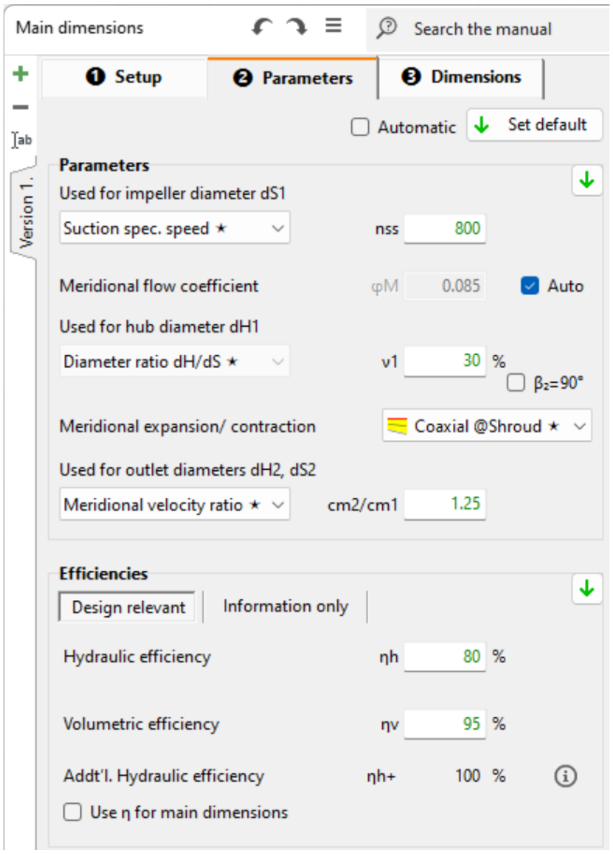
power fraction inducer

pressure rise inducer

power fraction radial impeller

pressure rise radial impeller

CFturbo 2025 calculates suitable parameters for the design of an optimum inducer at the design point and provides an estimate of important parameters such as shaft power or the expected efficiency.



☐ Automatic

Parameters

Used for impeller diameter dS1

Suction spec. speed ⋆ — nss 800

Meridional flow coefficient — φM 0.085 ☑ Auto

Used for hub diameter dH1

Diameter ratio dH/dS ⋆ — v1 30 %

☐ β₂=90°

Meridional expansion/ contraction — Coaxial @Shroud ⋆

Used for outlet diameters dH2, dS2

Meridional velocity ratio ⋆ — cm2/cm1 1.25

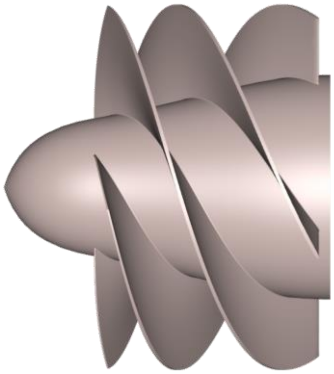
Efficiencies

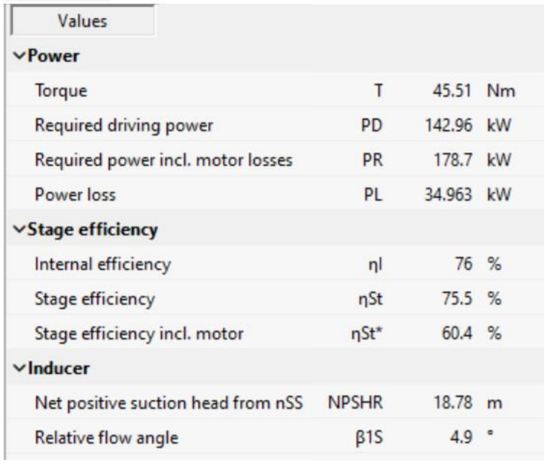
Hydraulic efficiency — ηh 80 %

Volumetric efficiency — ηv 95 %

Addt'l. Hydraulic efficiency — ηh+ 100 %

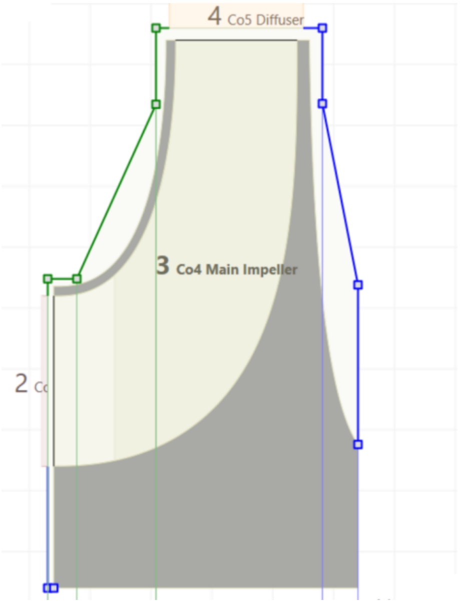
☐ Use η for main dimensions

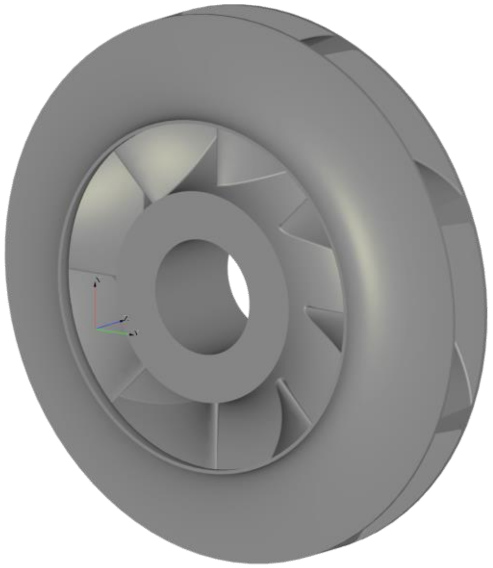




Values			
Power			
Torque	T	45.51	Nm
Required driving power	PD	142.96	kW
Required power incl. motor losses	PR	178.7	kW
Power loss	PL	34.963	kW
Stage efficiency			
Internal efficiency	ηI	76	%
Stage efficiency	ηSt	75.5	%
Stage efficiency incl. motor	ηSt*	60.4	%
Inducer			
Net positive suction head from nSS	NPSHR	18.78	m
Relative flow angle	β1S	4.9	°

The radial impeller is designed in the same way. In addition to the meridian contour, the entire impeller geometry can be modeled including the secondary flows.



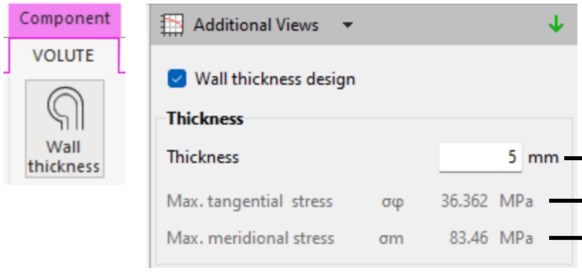


Creation of the Parameterized Base Geometry within CFturbo 2025

3D Modeling of the Radial Diffuser and the Volute

The flow from the radial impeller is discharged by a volute, the dimensions of which were also calculated by CFturbo 2025 to match the design point. CFturbo 2025 fills the radial gap between the two components with an unbladed radial diffuser. Bézier curves are used to give the spiral a somewhat compact, space-saving shape.

Particularly clever in CFturbo 2025 is the wall thickening of the volute for integration into a casing design. CFturbo 2025 facilitates the dimensioning of the wall thickness by calculating the maximum stresses in the material resulting from the pressurization.



☑ Wall thickness design

Thickness

Thickness 5 mm

Max. tangential stress σφ 36.362 MPa

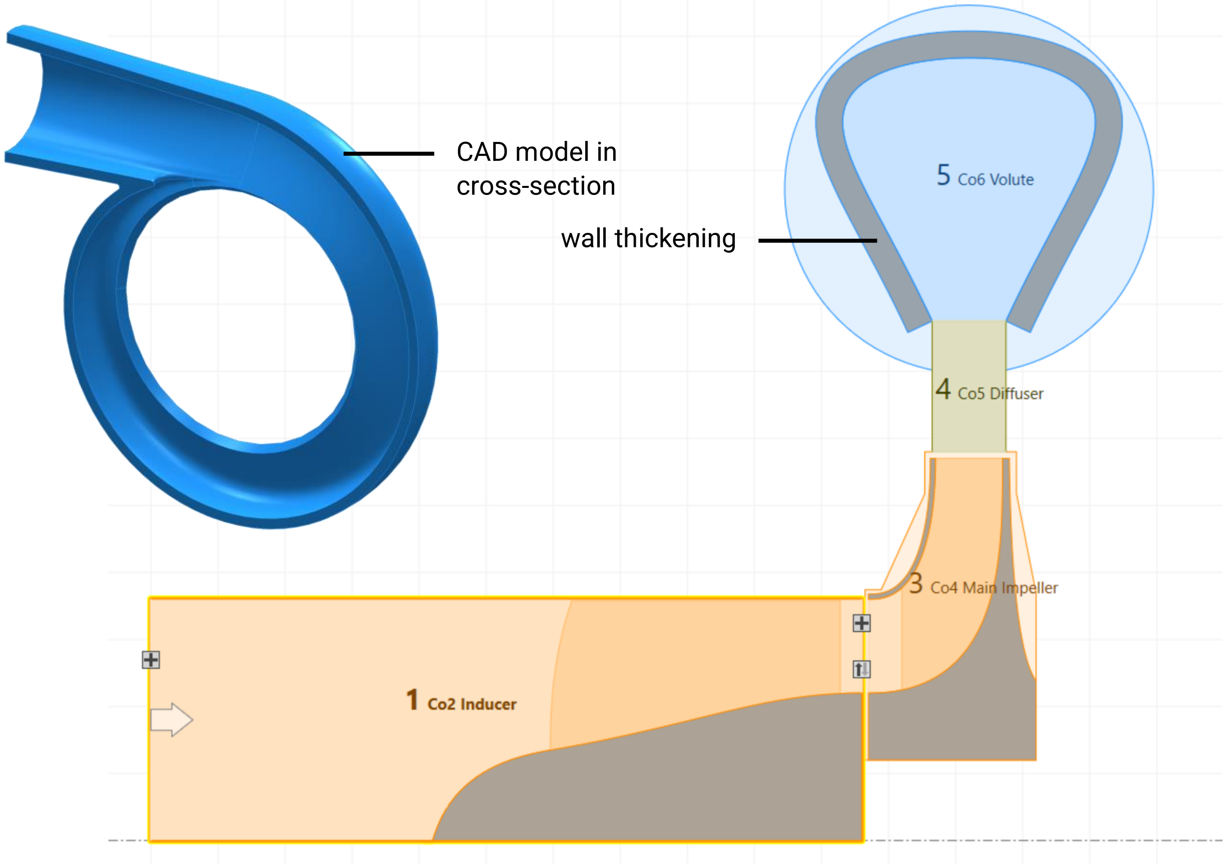
Max. meridional stress σm 83.46 MPa

defined wall thickness

calculated circumferential stress

calculated meridional stress

The overall model shown in the meridian section is now transferred to CFturbo BLADERUNNER 2025 to carry out the flow simulations.

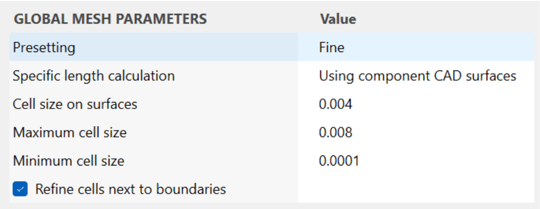


CAD model in cross-section

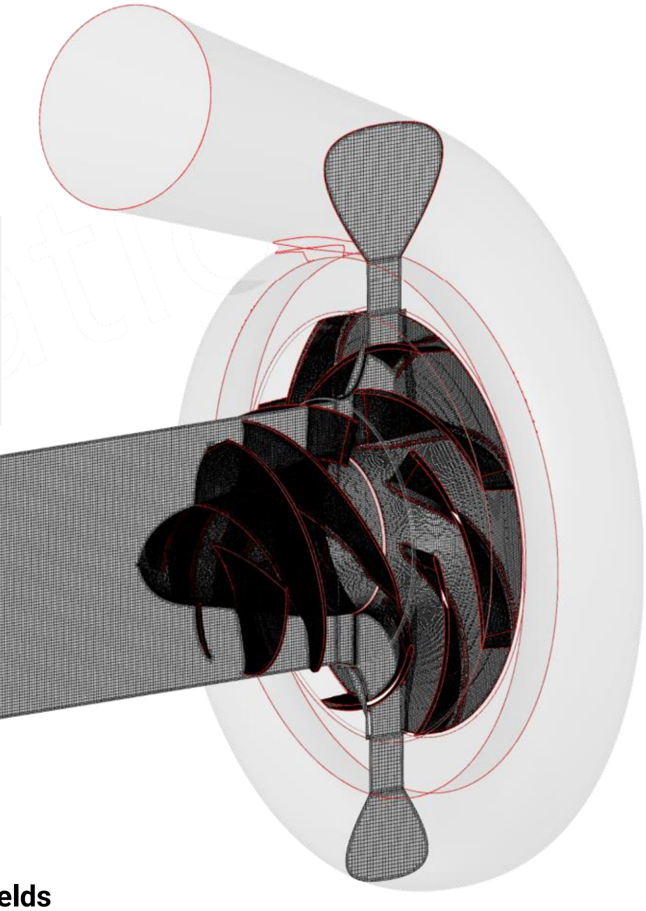
wall thickening

Calculation and Evaluation of the Flow Fields
Automatic Model Preparation and Meshing

To resolve the complex flow processes, the components in CFturbo BLADERUNNER 2025 are automatically meshed with the “fine” setting, and a computational grid with 4.7 million nodes is generated.

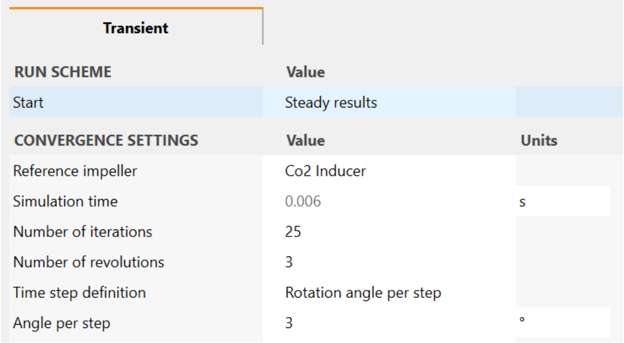


GLOBAL MESH PARAMETERS	Value
Presetting	Fine
Specific length calculation	Using component CAD surfaces
Cell size on surfaces	0.004
Maximum cell size	0.008
Minimum cell size	0.0001
☑ Refine cells next to boundaries	



Calculation and Evaluation of the Flow Fields
Transient Flow Simulations (CFD) Both Incompressible and with Cavitation

CFturbo BLADERUNNER 2025 offers the option of calculating flow fields with high temporal resolution using transient simulations in order to realistically map unsteady effects such as cavitation. To save computing time, it is often advisable to first carry out a steady-state flow calculation as a starting solution for the transient simulation in CFturbo BLADERUNNER 2025.

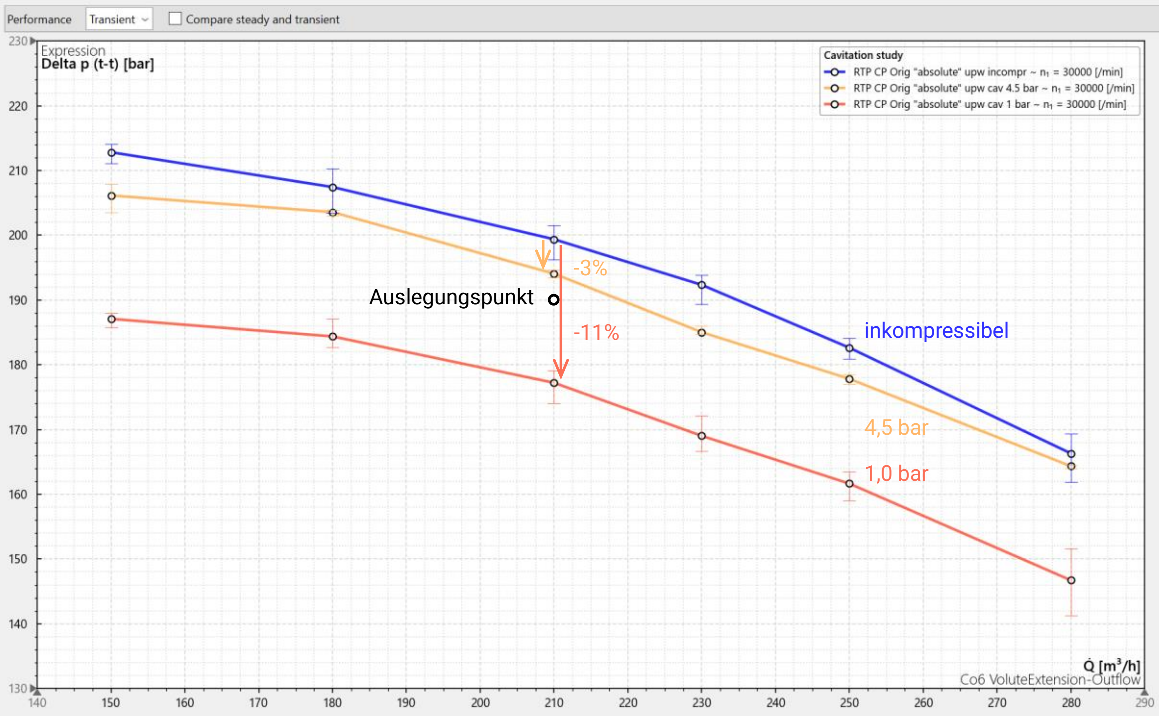


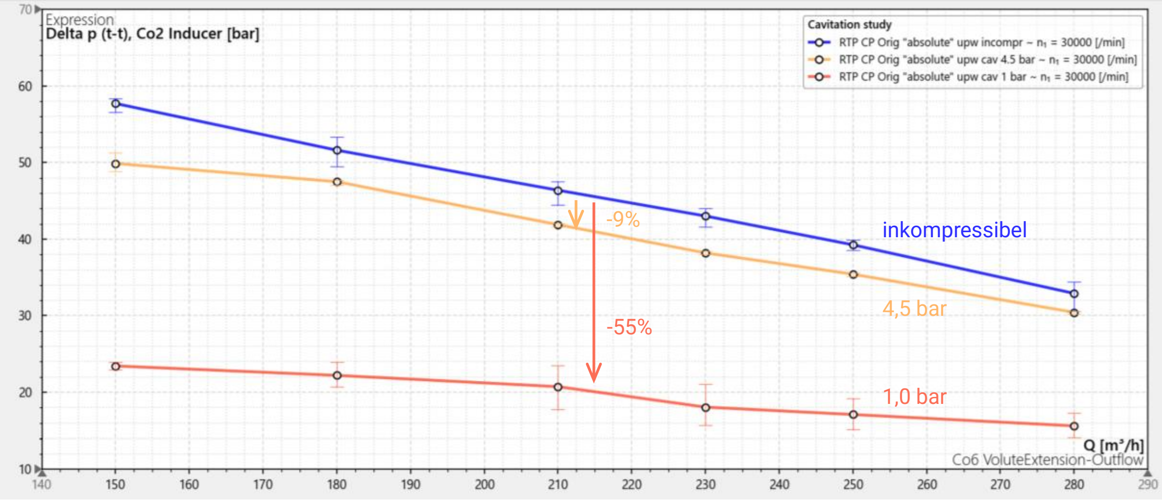
Transient

RUN SCHEME	Value	
Start	Steady results	
CONVERGENCE SETTINGS	**Value**	**Units**
Reference impeller	Co2 Inducer	
Simulation time	0.006	s
Number of iterations	25	
Number of revolutions	3	
Time step definition	Rotation angle per step	
Angle per step	3	°

Three transient simulations were carried out for the initial geometry: incompressible, with cavitation at $\Delta p_{t,\,LOX}$ = 1.0 bar and thus only just above the vapor pressure of $p_{S,\,LOX,\,-193°C}$ = 0.4 bar, and with cavitation at $\Delta p_{t,\,LOX}$ = 4.5 bar.

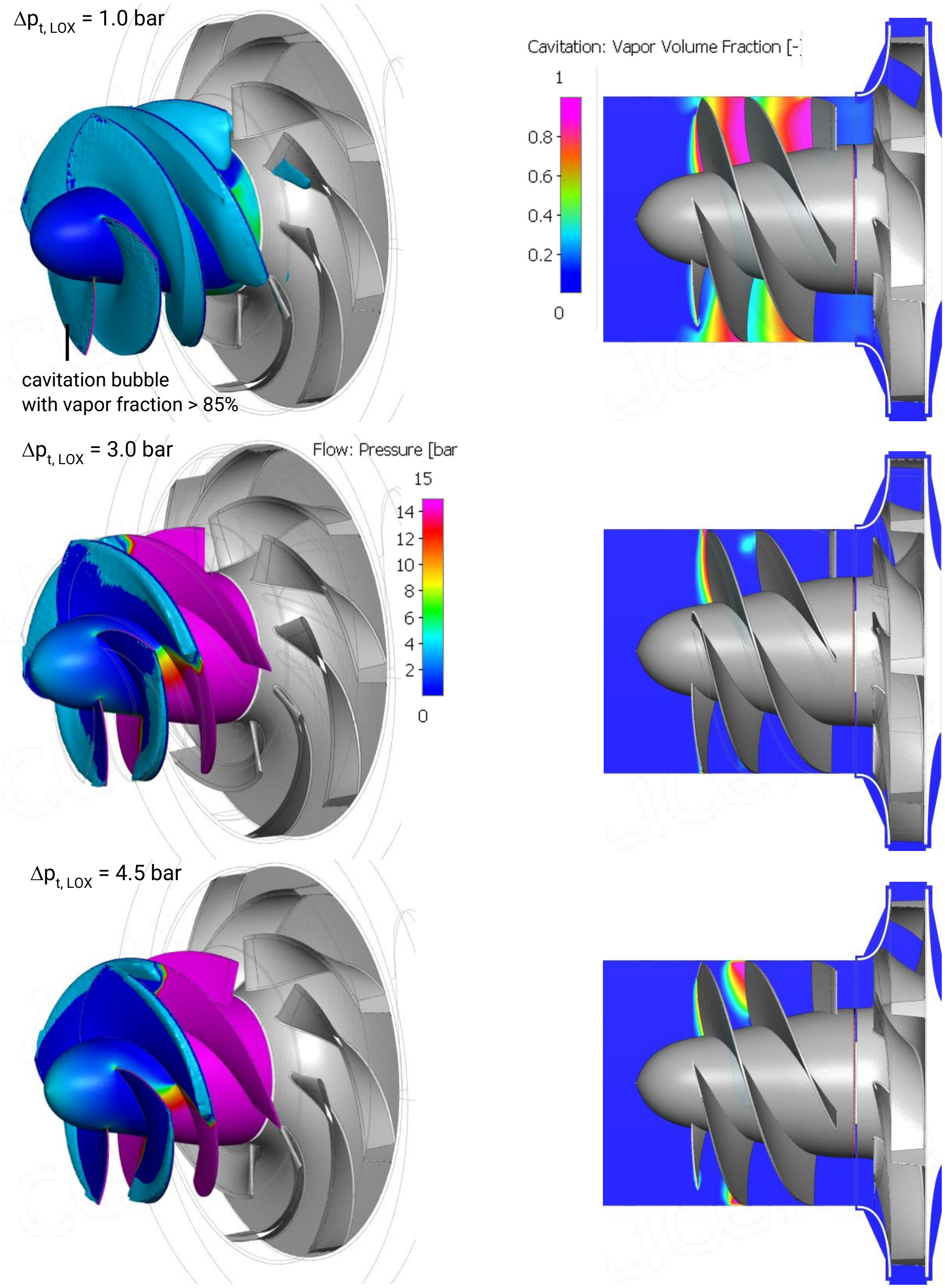
The diagrams show the characteristic curves for the total pressure increase $\Delta p_{t\text{-}t}$ in the complete stage (top) and only in the inducer (bottom). With incompressible simulation, the characteristic curve is clearly above the design point and can be regarded as the optimum case with the absence of cavitation due to a sufficiently high superimposed pressure.





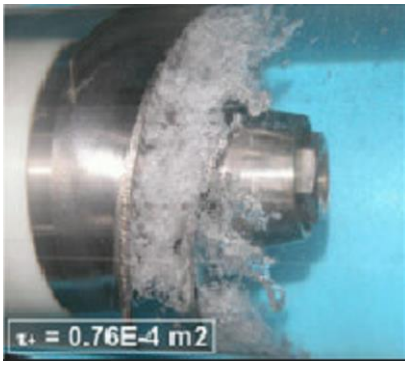
The occurrence of cavitation results in a disturbance of the flow field due to oxygen vapor bubbles. With the total pressure increase, there are moderate losses at $\Delta p_{t,\,LOX}$ = 4.5 bar, while $\Delta p_{t,\,LOX}$ = 1.0 bar, the loss is already very significant.

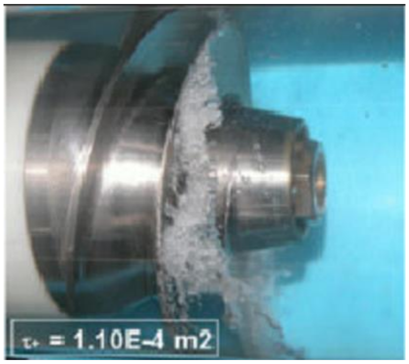
The following figures show the reduction in cavitation bubbles on the suction side of the inducer blading with increasing inlet pressure in a spatial view (left) and in a meridional section view (right).



At $\Delta p_{t,\,LOX}$ = 3.0 bar, the cavitation area is already so locally limited that safe operation of the centrifugal pump would be possible with a low total pressure loss. At $\Delta p_{t,\,LOX}$ = 4.5 bar, there is a high safety margin.

The results correspond very well with experimental studies such as [2]:



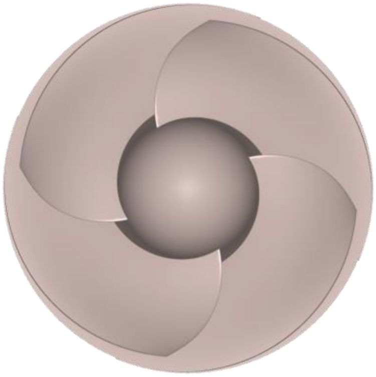


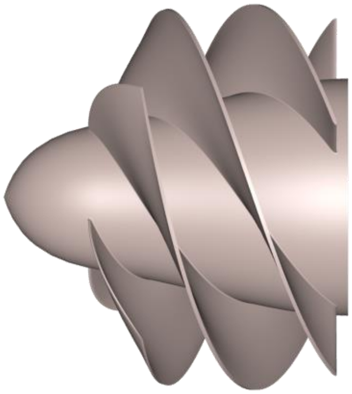
SUMMARY

The interaction between CFturbo 2025 and CFturbo BLADERUNNER 2025 enables the design and detailed analysis of a reliable LOX centrifugal pump in a single step. The decisive factor here is the new function in CFturbo BLADERUNNER 2025 to calculate flow fields transiently and thus to be able to map the cavitation process quantitatively with high accuracy.

OUTLOOK

To further reduce cavitation inducers are built with blade leading edges that are strongly curved against the direction of rotation and merge seamlessly into the blade tip [2]. As with the swept wing of an airplane, this reduces the flow velocity perpendicular to the leading edge. The blades are also tapered at the flow inlet. Such modeling with CFturbo 2025 is in progress …





REFERENCES

[1] https://en.wikipedia.org/wiki/Specific_impulse

[2] Luca d'Agostino: *Cavitation Experiments on Turbopump Inducers and Hydrofoils at Alta/Centrospazio*, ASME 2005 Fluids Engineering, 10.1115/FEDSM2005-77251