

SOFTWARE USER GUIDE

(Theory and general overview)

Computational Fluid x-Dynamics (CFxD)
(Version 2.14.5, October 31, 2026)

2026

Contents

	Nomenclature	3
	Introduction	5
I.	Theoretical background	6
	1.1 Governing equations	6
	1.2 Flow regimes	10
	1.3 Compressibility	12
	1.4 Unsteadiness	13
	1.5 Turbulence models	14
	1.6 Non-Newtonian fluids	18
	1.7 Heat transfer	20
	1.8 Acoustics	22
	1.9 Multiphase flow	23
	1.10 Combustion and reacting flow	25
	1.11 Porous resistance model	30
	1.12 Rotation modeling	31
	1.13 Mesh consideration	32
	1.14 Boundary conditions	33
	1.15 Limitations	36
II.	Getting started	38
	2.1 Installation	38
	2.2 Graphical User Interface (GUI)	40
	2.3 File structure	44
	2.4 History log	46
	2.5 Interactive help	47
	2.6 Licensing	48
	2.7 Typical workflows	49
III.	Geometry	55
	3.1 Mouse manipulation	56
	3.2 Geometry creation	57
	3.3 Geometry operation	68
	3.4 Geometry import	69
	3.5 Geometrical entities window	70
	3.6 View settings	71
	3.7 Naming	72
	3.8 Export geometry	73
IV.	Mesh	74
	4.1 Refinement	75
	4.2 Mesh statistics, Quality and Export	78
V.	Setup	80
	5.1 Physics model	81
	5.2 Run settings	84

	5.3 Operating conditions and buoyancy	86
	5.4 Domains	87
	5.5 Boundary conditions	89
VI.	Solution	94
	6.1 Stability control	95
	6.2 Monitoring point/face	96
	6.3 Start, Pause, Resume	97
VII.	Result	99
	7.1 Postprocessing at a point	100
	7.2 Postprocessing along a line	101
	7.3 Postprocessing on plane/boundary	102
	7.4 Streamlines and isosurfaces	104
	7.4 Deleting and hiding post entities	105
	7.5 Transient simulation result	107
	7.6 Report	108
	References	109

Nomenclature

Symbol	Dimension	Definition
c_i		model coefficients
c_p	$\frac{J}{(\mathrm{kg}\times\mathrm{K})}$	specific heat capacity by constant pressure
e_{ijk}		Levi-Civita tensor
g_i	$\frac{m}{s^2}$	gravity acceleration
Kn		Knudsen number
k	$\frac{m^2}{s^2}$	turbulent kinetic energy
l	m	characteristic length scale
M		Mach number
m	kg	mass
Nu		Nusselt number
P_k		turbulent production
Pr		Prandtl number
Pr_t		Prandtl turbulent number
p	Pa	static pressure
R	$\frac{J}{(\mathrm{kg}\times\mathrm{K})}$	gas constant
Re		Reynolds number
Sc		Schmidt number
S_{ij}	$\frac{1}{s}$	mean strain-rate tensor (components)
T	K	temperature
t	s	time
u, u_i	$\frac{m}{s}$	velocity vector/components
$\overline{u_i' u_j'}$	$\frac{m^2}{s^2}$	Reynolds stress tensor (components)
$\overline{u_j' \phi'}$		scalar flux (components)
V	m^3	volume
x_i	m	cartesian coordinates

Greek letters

Symbol	Dimension	Definition
$\alpha_i, \beta_i, \delta_i, \gamma_i$		model coefficients
$\dot{\gamma}$		strain rate

Γ_ϕ		Molecular diffusion coefficient of a general scalar quantity, ϕ
δ_{ij}		Cartesian components of unit tensor (Kronecker delta)
ε	$\frac{m^2}{s^2}$	dissipation rate of turbulent kinetic energy
λ	$\frac{J}{(\mathrm{s\times K\times m})}$	thermal conductivity
ϕ		general scalar quantity
κ		Karman constant
μ, μ_t	$\frac{\mathrm{kg}}{(\mathrm{m\times s})}$	dynamic molecular/turbulent viscosity
ν, ν_t	$\frac{m^2}{s}$	kinematic molecular/ turbulent viscosity
ρ	$\frac{\mathrm{kg}}{m^3}$	density
σ	m^2	surface
σ_ϕ		turbulent Schmidt (or Prandtl) number for variable
τ	s	turbulent time scale
τ_{ij}		Reynolds stress tensor (components)
ω		specific dissipation rate

Abbreviations

Abbreviation	Definition
CAD	Computer-aided design
CDS	Central Difference Scheme
CFD	Computational Fluid Dynamics
CV	Control Volume
DNS	Direct Numerical Simulation
EVM	Eddy-Viscosity Model
LES	Large Eddy Simulation
RANS	Reynolds Averaged Navier-Stokes
RSM	Reynolds Stress Model
RST	Reynolds Stress Tensor
SIMPLE	Semi Implicit Method for Pressure Linked Equations
SIP	Strongly Implicit Procedure
UDS	Upwind Difference Scheme
URANS	Unsteady RANS

Introduction

Computational Fluid x-Dynamics (CFxD) is a software application designed to model and simulate fluid flow. It includes the complete workflow for Computational Fluid Dynamics (CFD) analysis, from geometry creation to result visualization, in a streamlined and user-friendly interface. The typical simulation pipeline in CFxD includes:

Geometry → Mesh → Setup → Solution → Result.

CFxD allows users to create geometrical models from scratch or import existing geometry, generate computational meshes, define physical models and boundary conditions, solve the flow field using robust solvers, and visualize simulation results.

The software features a simplified and intuitive graphical interface, designed to provide a convenient tool for engineers without extensive experience in CFD. The current version supports **steady single-phase flow** in both **laminar and turbulent regimes**. It supports **incompressible** and **compressible** flow, as well as **heat transfer**, including **conjugate heat transfer**.

CFxD functions as a graphical shell that leverages free and open-source libraries, including:

- Open CASCADE Technology (OCCT) for CAD and geometry operations
- OpenFOAM for mesh generation and flow simulation
- VTK (Visualization Toolkit) for post-processing and result visualization.

The application has been tested on **Windows 10/11** platform. Hardware requirements depend on the complexity of the CAD geometry and mesh density. CFxD supports multi-core CPUs to accelerate computation.

CFxD is distributed **"as is"**, without any warranties of any kind. The user assumes full responsibility and risk for its use. All product names, logos and brands are the properties of their respective owners.

Chapter I. Theoretical background

CFxD is based on the principles of **Computational Fluid Dynamics (CFD)**, which involves solving the governing equations of fluid motion using numerical methods. This section provides a brief overview of the physical models and numerical techniques used in CFxD.

1.1 Governing equations

Mass conservation equation

The **integral form** of the continuity equation (mass conservation) over a fixed control volume V with enclosing surface S is

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho dV}_{I} + \underbrace{\int_\sigma \rho\vec{u}\,\vec{n}d\sigma}_{II} = 0\,, \tag{1}$$

where ρ is fluid density, $\vec{u}$ is velocity vector and $\vec{n}$ is the outward-pointing unit normal vector on the surface S. This equation states that the rate of mass change inside the volume (Term I) is equal to the net mass flux across the surface (Term II).

Applying **Gauss divergence theorem**, the surface integral is converted to a volume integral:

$$\frac{\partial}{\partial t}\int_V \rho dV + \int_V div(\rho\vec{u})dV = 0\,. \tag{2}$$

The corresponding **differential form** of the continuity equation is

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0\,. \tag{3}$$

For **incompressible flow** ($\rho = const$) the continuity equation reduces to

$$\frac{\partial u_i}{\partial x_i} = 0\,. \tag{4}$$

Momentum conservation equation

The **integral form** of momentum conservation equation is

$$\frac{\partial}{\partial t}\int_V \rho\vec{u}dV + \int_S \rho\vec{u}(\vec{u}\cdot\vec{n})dS = \int_S \vec{\tau}\cdot\vec{n}dS + \int_V \rho\vec{g}dV, \tag{5}$$

where $\vec{g}$ is the gravity acceleration vector and τ is the stress tensor, which includes both pressure and viscous contributions.

Applying Gauss divergence theorem to the surface integrals and assuming the control volume becomes infinitely small, the differential form of the momentum conservation equation becomes:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i. \tag{6}$$

For Newtonian fluids, the stress tensor τ_{ij} is modeled using the Stokes hypothesis [9] as

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\mu\frac{\partial u_k}{\partial x_к}\delta_{ij}, \tag{7}$$

where is μ dynamic viscosity and δ_{ij} is Kronecker delta.

Taking into account (7), the equation for the conservation law of momentum can be written in the following form:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\mu\frac{\partial u_k}{\partial x_к}\delta_{ij}\right] + \rho g_i\,. \tag{8}$$

This equation is known as Navier-Stokes equation. For incompressible flows with constant viscosity, Navier-Stokes equation simplifies to

$$\frac{\partial u_i}{\partial t} + u_j\frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\mu}{\rho}\frac{\partial^2 u_i}{\partial x_i \partial x_j} + g_i. \tag{9}$$

For flow in a rotating reference frame, the Coriolis $\left(-2\vec{\Omega}\times\vec{u}\right)$ and the centrifugal acceleration $\left(-\vec{\Omega}\times\vec{\Omega}\times\vec{x}\right)$ must be added:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p^*}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + f_i^*, \tag{10}$$

where the quantity pressure p^* is defined as

$$p^* = p - \frac{1}{2}\rho[(x_i x_i)(\Omega_k\Omega_k) - (x_k\Omega_k)^2]. \tag{11}$$

The body-force term including the Coriolis contribution is

$$f_i^* = f_i - 2\rho e_{ijk}\Omega_j u_k, \tag{12}$$

where e_{ijk} is the Levi-Civita tensor.

Energy, scalar, and species conservation equation

The **integral form** of scalar transport equation is

$$\frac{\partial}{\partial t}\int_V \rho\phi dV + \int_S \rho\phi\vec{u}\cdot\vec{n}dS = \int_S D_\phi \cdot \vec{n}\, dS + \int_V S_\phi dV. \tag{13}$$

The diffusive flux D_ϕ can be modeled using Fourier's law, which relates the flux to the gradient of a scalar. Applying Gauss divergence theorem and and assuming the control volume becomes infinite small, the **differential form** of the scalar transport equation is:

$$\frac{\partial\phi}{\partial t} + \frac{\partial\phi u_i}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\Gamma_\phi \frac{\partial\phi}{\partial x_i}\right) + S_\phi. \tag{14}$$

The diffusion coefficient Γ_ϕ is generally expressed as a ratio of kinematic viscosity ν to Prandtl number Pr :

$$\Gamma_\phi = \frac{\nu}{Pr}. \tag{15}$$

Prandtl number is often used in the heat transfer calculations. It characterizes the ratio of momentum and thermal diffusivities in fluid. The typical values for diatomic gases are $Pr \geq 0.72$ and for triatomic and polyatomic are $0.75 \leq Pr \leq 1$.

Substituting $\phi = c_p T$, the scalar transport equation becomes the conservation of energy in differential form

$$\rho\frac{\partial c_p T}{\partial t} + \frac{\partial u_i c_p T}{\partial x_j} = -\frac{\partial q_i}{\partial x_i} + \frac{\partial \tau_{ij} u_i}{\partial x_j} + S\,, \tag{16}$$

where $q_i = -k\frac{\partial T}{\partial x_i}$ defined by Fourier's law.

Gas mixture can be modeled with a transport equation for the species mass fractions. The mass fraction of species i is defined as

$$Y_i = \frac{m_i}{m}, \tag{17}$$

where m_i is the mass of species and m is the total mas of the gas mixture. The sum of all species mass fractions is equal to one:

$$\sum_{i=1}^{N} Y_i = 1. \tag{18}$$

For a non-reacting gas mixture, the conservation equation for species i can be written in differential form as

$$\frac{\partial\rho Y_i}{\partial t} + \frac{\partial\rho u_j Y_i}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\rho D_{i,e}\frac{\partial Y_i}{\partial x_j}\right), \tag{19}$$

where Y_i is the mass fraction of species i, $D_{i,e}$ is the effective mass diffusivity, and ρ is the mixture density.

Equation of state

To **close the system of governing equations**, an additional relationship is required to connect thermodynamic variables such as **pressure**, **temperature**, and **density**. This is provided by an **equation of state** (EOS), which relates to fluid properties. A commonly used EOS for gases is the **ideal gas law**:

$$p = \rho RT, \tag{20}$$

where p is pressure, ρ is density, R is the specific gas constant and T is temperature. The ideal gas law assumes that gas molecules do not interact with each other and occupy no volume. It is valid for many engineering applications.

1.2 Flow regimes

CFxD supports the simulation of both laminar and turbulent flows. The term *laminar* originates from the Latin word "*lamina"*, meaning a plate or a layer. In laminar flow, fluid moves in parallel layers with minimal mixing between them, typically at **low velocities**. An example of laminar flow through a channel is shown in Figure 1, where smoke added to the air helps visualize the smooth flow paths.

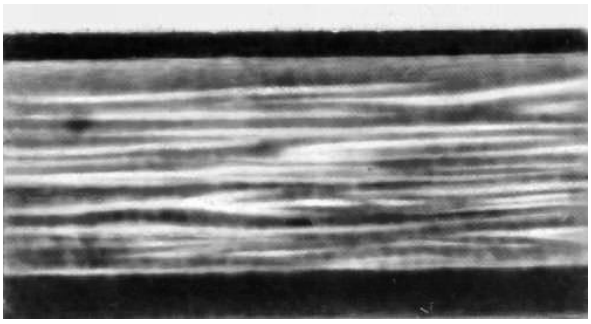


Figure 1. Laminar flow in a channel.

As the flow speed increases, the fluid layers begin to interact and mix. This intermediate state is known as **transitional flow**. Eventually, the flow may become **fully turbulent**, characterized by chaotic, irregular motion and strong mixing (Figure 2).

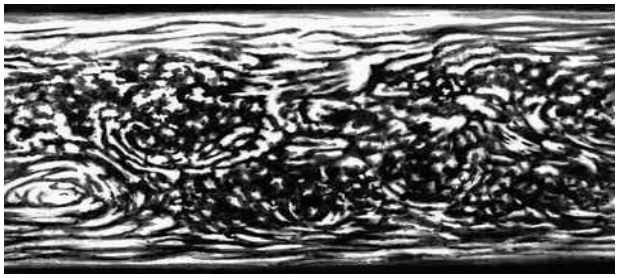


Figure 2. Turbulent flow in channel.

The flow regime can be characterized by Reynolds number, defined as

$$Re = \frac{l_0 \cdot u_0}{\nu}, \tag{21}$$

where l_0 is a characteristic length, u_0 is a characteristic (bulk) velocity and ν is a kinematic viscosity of fluid. The *Reynolds number* expresses the ratio between inertial and viscous (or molecular) forces. If this ratio is small, the viscous (or molecular) forces are comparable to the inertial forces, and the flow keeps its regular structure (laminar flow). If the ratio becomes larger, the viscous forces do not suffice to compensate the inertial forces. The flow turns unstable, and small initial perturbations destroy the regular flow structure, causing turbulence

(turbulent flow). For internal flow, such flow in a pipe, the transition to turbulence typically occurs around $Re > 2300$.

1.3 Compressibility

CFxD supports both **incompressible** ($\rho = const$) and **compressible** ($\rho \neq const$) flow simulations. These terms refer to whether the **fluid density** varies within the fluid:

- A flow is considered **incompressible** when the fluid density remains approximately the same
    - For example, **water** is typically modeled as incompressible because its density changes very little under pressure.
    - **Air** can also be treated as incompressible **at low velocities**, where density variation is negligible.
- A flow is **compressible** when changes in pressure or temperature cause **significant variations in fluid density**. This is typical at **high velocities**, such as in gas dynamics or supersonic flows.

The degree of compressibility is often characterized by **Mach number**

$$M = \frac{u}{a}, \tag{22}$$

where u is flow velocity and a is speed of sound in the medium. At low speeds $M < 0.3$, the flow is generally considered incompressible. At high speeds, $M \geq 0.3$, the compressibility effects begin to play a significant role. At $M > 1$, the flow becomes supersonic and special phenomena such as shock waves may occur. An example of compressibility is shown in Figure 3, where a sphere moving at high speed creates a **shock wave** — a sharp pressure discontinuity caused by the fluid being compressed faster than disturbances can propagate.

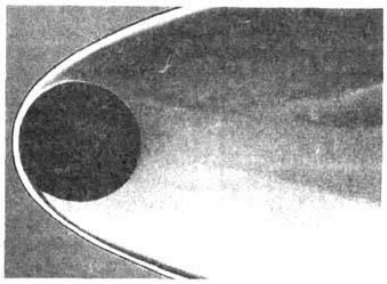


Figure 3. A shock wave in front of a sphere moving at a high speed.

Similar shock waves may occur in internal flows, such as nozzles or channels, when the flow velocity exceeds the local speed of sound. In such cases, mesh refinement near the shock is often necessary to accurately resolve the steep gradients in pressure, temperature, and velocity**.**

1.4 Unsteadiness

CFxD supports both **steady** and **unsteady** flow simulations. All flows are **unsteady** - they change with time. However, if the flow reaches a statistically stable pattern or the time-dependent effects are negligible, it can be approximated as **steady**, meaning the flow variables do not vary with time, t:

- **Steady flow**: $\frac{\partial}{\partial t} = 0$
- **Unsteady flow**: $\frac{\partial}{\partial t} \neq 0$

Choosing between steady and unsteady modeling depends on the physical nature of the problem and the level of accuracy required. For example:

- Flow through a long straight pipe at constant inlet conditions can often be treated as steady.
- Vortex shedding behind a cylinder or pressure fluctuations in a combustion chamber require unsteady modeling.

1.5 Turbulence models

CFxD supports two commonly used turbulence models:

- **k–ε model**
- **Shear Stress Transport (SST) model**

Both are based on the **Reynolds-Averaged Navier–Stokes (RANS)** approach, which simplifies turbulent flows by averaging out high-frequency fluctuations in time. This allows efficient simulation of turbulent flows without resolving all the small-scale structures, as would be required in **Direct Numerical Simulation (DNS)** or **Large Eddy Simulation (LES)**.

Reynolds-Averaged Equations

In the RANS framework, each variable (e.g., velocity, pressure, temperature) is decomposed into:

$$u_i = \overline{u}_i + u_i', \tag{23}$$

where $\overline{u}_i$ is the time-averaged (mean) velocity and u_i' is the fluctuating component. This decomposition results in the **averaged equations** for incompressible flow:

- **Continuity**:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0, \tag{24}$$

- **Momentum**:

$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial \overline{u_i u_j}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} - \frac{\partial}{\partial x_j}\left(\nu \frac{\partial \bar{u}_i}{\partial x_i} + \frac{\partial \overline{u_i' u_j'}}{\partial x_j}\right) + g_i. \tag{25}$$

The term $\overline{u_i' u_j'}$ is the **Reynolds stress tensor**, which introduces new unknowns. These need to be modeled using turbulence models — a challenge known as the **closure problem**. Most of turbulence models used in the engineering applications are based on the concept of eddy viscosity [9]. Reynolds stress tensor can be represented as a product of eddy viscosity ν_t and mean strain-rate tensor $\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)$:

$$-\overline{u_i' u_j'} = \nu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) - \frac{2}{3} k \delta_i, \tag{26}$$

where $k = \frac{1}{2}\overline{u_i' u_i'}$ is a turbulent kinetic. The given equation does not represent a model of turbulence but only characterizes the structure of such a model. So, the basic problem is to find the expression of the turbulent viscosity ν_t. It can be done by solving two differential equations as shown below.

k–ε Turbulence Model

The **standard k–ε model** introduces two additional transport equations:

- **k**: turbulent kinetic energy
- **ε**: turbulent dissipation rate.

The model equations are:

Turbulent kinetic energy:

$$\rho \frac{\partial k}{\partial t} + \rho \overline{u}_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial \overline{u}_i}{\partial x_j} - \rho \varepsilon + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right]. \quad (27)$$

Dissipation rate:

$$\rho \frac{\partial \varepsilon}{\partial t} + \rho \overline{u}_j \frac{\partial \varepsilon}{\partial x_j} = c_{\varepsilon 1} \frac{\varepsilon}{k} \tau_{ij} \frac{\partial \overline{u}_i}{\partial x_j} - c_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right]. \quad (28)$$

The **eddy viscosity** is computed as:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}. \quad (29)$$

The model's constants are

$$c_{\varepsilon 1} = 1.44; c_{\varepsilon 2} = 1.92; C_\mu = 0.09; \sigma_k = 1.0; \sigma_\varepsilon = 1.3.$$

The k–ε model is widely used due to its **robustness** and **simplicity**, but it tends to underperform in flows with **strong separation**, **swirl**, or **adverse pressure gradients**.

SST (Shear Stress Transport) Model

The **SST model** blends the advantages of the:

- **k–ω model** near walls, and
- **k–ε model** in the free stream

This hybrid approach improves accuracy in **boundary layer separation** and **near-wall treatment**. The SST model uses the following transport equations:

- Turbulent kinetic energy (k)
- Specific dissipation rate (ω).

The basic *two-layer mode*l of Menter is presented below:

$$\rho \frac{\partial k}{\partial t} + \rho \overline{u}_j \frac{\partial k}{\partial x_j} = \widetilde{P_k} - \beta^* \rho k \omega + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right], \quad (30)$$

$$\rho\frac{\partial\omega}{\partial t}+\rho\overline{u}_j\frac{\partial\omega}{\partial x_j}=\alpha\rho S-\beta\rho\omega^2+\frac{\partial}{\partial x_j}\left[(\mu+\sigma_\omega\mu_t)\frac{\partial\omega}{\partial x_j}\right]+$$
$$+2(1-F_1)\rho\sigma_{\omega2}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j}, \quad (31)$$

The turbulent viscosity μ_t and Reynolds stresses τ_{ij} are defined similar as in $k-\omega$ *model*. Denoting the generalized parameter φ_1 with a set of constants of the original model $k-\omega$ with index 1 and accordingly φ_2 with a similar set of constants of a transformed $k-\varepsilon$ *model*, the blending function can be written as

$$\varphi=F_1\varphi_1+(1-F_1)\varphi_2. \quad (32)$$

The blending function is determined as follows:

$$F_1=tanh(arg_1^4), \quad (33)$$
$$arg_1=min\left[max\left(\frac{\sqrt{k}}{\beta^*\omega y};\frac{500\upsilon}{y^2\omega}\right);\frac{4\rho\sigma_{\omega2}k}{CD_{k\omega}y^2}\right], \quad (34)$$
$$CD_{k\omega}=max\left\{2\rho\sigma_{\omega2}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j},10^{-10}\right\}. \quad (35)$$

The turbulent eddy viscosity is defined as

$$\nu_t=\frac{a_1k}{max(a_1\omega,SF_2)}, \quad (36)$$

where S is the strain rate, F_2 is second blending function defined similar to (30).

$$F_2=tanh(arg_2^2), \quad (37)$$
$$arg_2=max\left[2\sqrt{k}/0{,}09\omega y;\frac{500\nu}{y^2\omega}\right]. \quad (38)$$

The production limiter $\widetilde{P_k}$ is defined as

$$\widetilde{P_k}=\min(\tau_{ij}\frac{\partial\overline{u}_i}{\partial x_j},10\beta^*\rho k\omega). \quad (39)$$

The following constants are used:

1. Model constant of $k-\omega$ *model*

$$\beta^*=0.09;\alpha_1=\frac{5}{9};\beta_1=\frac{3}{40};\sigma_{k1}=0.85;\sigma_{\omega1}=0.5;$$

2. Model constant of $k-\varepsilon$ *model*

$$\beta^*=0.09;\alpha_2=0.44;\beta_2=0.0828;\sigma_{k2}=1;\sigma_{\omega2}=0.856;$$

Turbulent scalar/species transport

For turbulent flows, the effective diffusivity includes molecular and turbulent contributions in equation (19):

$$D_{i,e} = D_i + \frac{\nu_t}{Sc_t}, \tag{40}$$

where D_i is the molecular diffusivity, ν_t is the turbulent kinetic viscosity, and Sc_t is the turbulent Schmidt number. For a binary mixture, only one species equation is required because the second mass fraction is obtained from

$$Y_2 = 1 - Y_1, \tag{41}$$

This reduces the number of solved scalar equations and ensures conservation of the total mixture composition. For ideal-gas mixtures, the molecular weight mixture is calculated from the species mass fractions as

$$\frac{1}{M_{mix}} = \sum_{i=1}^{N} \frac{Y_i}{M_i}, \tag{42}$$

where M_{mix} is the molecular weight of species i. The mixture density is then calculated using the ideal-gas equation of state.

1.6 Non-Newtonian fluids

The shear stress was linearly proportional to strain rate in equation (7). Such fluids are called *Newtonian*. Many liquids or gases can be considered as Newtonian, whereas there are many other materials that do not satisfy the linearity. Such liquids are called *non-Newtonian fluids*. Rheology studies such fluids. In rheology, the equation (7) is re-written in the following form:

$$\tau = \mu \frac{du}{dy} = \mu \dot{\gamma}, \tag{43}$$

where $\dot{\gamma}$ is the strain rate. The viscosity μ in case of non-Newtonian fluids depends not only on temperature and pressure, similar to Newtonian fluids, but on the strain rate, pre-history, and time.

According to their properties, *non -Newtonian fluids* can be classified into the following types:

1. *Shear-thinning fluids* or *pseudo-plastic*: In such fluids, the viscosity decreases when the strain rate increases. Such fluids are widely used. Examples are paints, emulsions, suspensions, etc.
2. *Shear-thickening fluids* or *dilatants*: In these fluids, in contrast to that observed in the first group, the viscosity increases when the strain rate increases. These fluids are not used widely. Some examples are syrups, different suspensions, mixture of sand and water, etc.
3. *Viscoplastic*. Such fluids are similar to solid in appearance, but under force, their behavior changes to liquid. Under critical stress, they may behave as non-Newtonian fluids (Bingham plastic), i.e., shear-thinning or shear-thickening fluids. Ketchups, gels, toothpaste, etc. belong to this group. A fluid diagram with different non-Newtonian fluids is shown in Figure 4. For comparison, Newtonian fluids are also demonstrated.

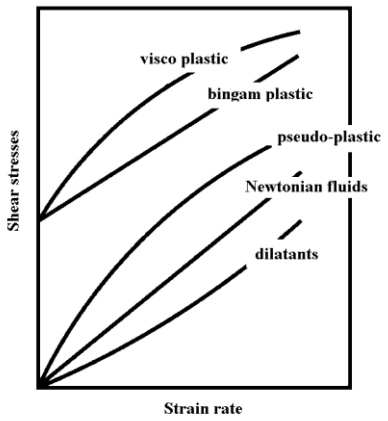


Figure 4. The fluid classification.

Another type of non-Newtonian fluids includes fluids in which viscosity is a function of time at the constant ambient conditions and strain rate. If viscosity decreases in time, such fluids are called thyrotrophic; under, the opposite conditions, it is anti-thyrotrophic or rheopectic. In reality, the fluids often have several properties simultaneously.

The commonly used law to describe the shear-thinning fluids is a *power law*, which was suggested by Oswald de Waele :

$$\tau = m\dot{\gamma}^n, \tag{44}$$

where n is an exponent and m is the flow consistency index. Both constants depend on fluid properties. The exponent n shows how much further a fluid is from being a Newtonian fluid. At $n = 1$, the fluid is Newtonian. The flow consistency index is proportional to the fluid viscosity. Equation 44 can be re-written for viscosity as

$$\mu = m\dot{\gamma}^{n-1}. \tag{45}$$

Power law can be used for dilatants as well. In this case, the exponent is greater than 1, i.e., $n > 1$. Despite the wide use of power law to describe non-Newtonian fluids, this law does not describe the extreme cases at $\dot{\gamma} \to 0$ and $\dot{\gamma} \to \infty$. In these cases, for pseudo-plastic fluids, the plateau zone can be observed. The appearance of the plateau zone for dilatants is still not well described.

1.7 Heat transfer

Heat transfer in fluids occurs through the following mechanisms:

- **Convection** – transport of thermal energy by fluid motion,
- **Conduction** – molecular diffusion of heat through the fluids or solids,
- **Radiation** – energy transfer through electromagnetic waves.

CFxD supports **convection** and **conduction** in **single-phase** flows, including **conjugate heat transfer (CHT)**, where heat conduction in solid regions is coupled with heat transfer in adjacent fluids regions. **CFxD** also supports a **surface-to-surface** (**S2S**) radiation model for radiative heat exchange between wall surfaces.

CFxD computes temperature by solving an **energy (thermal) transport equation**. An additional turbulent heat-flux contribution is modeled through an effective thermal diffusivity. CFxD also supports **buoyancy** (natural convection): gravity introduces a density-driven body force in the momentum equation when temperature-dependent density effects are enabled. As result, CFxD can simulate **forced convection**, **conduction**, **buoyancy-driven** flow, **conjugate heat transfer** and **surface-to-surface** radiative heat exchange.

The general form of the energy (temperature) transport equation is

$$\frac{\partial \overline{\phi}}{\partial t} + \overline{u}_j \frac{\partial \overline{\phi}}{\partial x_j} = \frac{\partial}{\partial x_j}\left(D \frac{\partial \overline{\phi}}{\partial x_j} - \overline{u_j' \phi'} \right) + \bar{S}_\phi, \tag{46}$$

where $\overline{\phi}$ is the mean scalar (temperature), $\bar{S}_\phi$ represents a source term of the transported scalar and $-\overline{u_j' \phi'}$ is the turbulent scalar flux term. The turbulent scalar flux is commonly modelled using a gradient-diffusion approach [9]:

$$-\overline{u_j' \phi'} = \frac{\nu_t}{\sigma_\varphi} \frac{\partial \overline{\phi}}{\partial x_j}, \tag{47}$$

where σ_φ is the turbulent Schmidt/Prandtl number, depending on the scalar.

Surface-to-surface radiation

The **surface-to-surface (S2S)** radiation model is used to calculate radiative heat exchange between solid wall surfaces. It is suitable for cases where the participating medium is transparent, meaning radiation absorption, emission, and scattering inside the fluid volume are neglected. Therefore, radiation is exchanged only between surfaces.

In the S2S model, wall surfaces are treated as **gray diffuse surfaces**. The radiative heat exchange depends on the surface temperature, emissivity, and geometric visibility between surfaces. The emitted radiative heat flux from the surface is proportional to the fourth power of its absolute temperature:

$$q_{\text{emit}} = \varepsilon \sigma T^4, \tag{48}$$

where ε is the surface emissivity, σ is the Stefan–Boltzmann constant, and T is the absolute surface temperature. The net radiative heat exchange also depends on view factors between surfaces, which describe how much of the radiation leaving one surface reaches another surface [9].

In CFxD, S2S radiation is applied on **wall boundaries**. The user specifies the wall emissivity, while the radiation model calculates the radiative exchange between participating wall surfaces. This model is appropriate for enclosure-type problems, heated cavities, thermal boxes, and other applications where surface radiation is important and the medium between surfaces can be considered transparent.

The S2S model does not account for participating-media radiation, such as absorption or emission by gases, soot, particles, droplets, or combustion products. For such cases, more advanced radiation models would be required.

1.8 Acoustics

Acoustic prediction is relevant to engineering applications in which turbulent flow generates broadband noise, including fans, pumps, ducts, heat exchangers, turbomachinery, and other flow systems. In such applications, the acoustic energy is often distributed over a broad frequency range rather than concentrated at distinct tonal frequencies.

CFxD provides a simplified broadband acoustic model based on the Proudman analogy. The model estimates the acoustic power generated by isotropic turbulence from the mean-flow and turbulence quantities obtained during the flow simulation. It is intended for estimating turbulent noise-source strength and comparing alternative designs; it does not solve the propagation of acoustic waves or predict the sound pressure level at a microphone.

The general Proudman formulation expresses the acoustic power density as

$$P_A = a\rho_0 \left(\frac{u^3}{l}\right) \frac{u^5}{a_0^5}, \tag{49}$$

where P_A is the acoustic power generated per unit volume, a is a model constant, ρ_0 is the reference density, u is a characteristic turbulent velocity, l is a characteristic turbulent length scale, and a_0 is the reference speed of sound.

For turbulence models that provide the turbulent kinetic energy k and the dissipation rate ε, the formulation is written in the form used by CFxD:

$$P_A = a_\varepsilon \rho \varepsilon \left(\frac{\sqrt{2k}}{a_0}\right)^5, \tag{50}$$

where ρ is the local fluid density, k is the turbulent kinetic energy, ε is the turbulent kinetic-energy dissipation rate, and a_0 is the local or reference speed of sound. CFxD uses $a_\varepsilon = 0.1$.

1.9 Multiphase flow

Multiphase flows contain two or more fluids that occupy the same computational domain while remaining separated by moving or deformable interfaces. Examples include liquid–gas flows, filling and draining, sloshing, wave motion, stratified flow, droplet transport, and the interaction of immiscible liquids. In these problems, the location of the interface changes during the simulation. Resolving the interface is therefore different from solving a single-phase flow, because the local material properties depend on the amount of each phase present in a computational cell.

CFxD models two immiscible Newtonian fluids using the **Volume-of-Fluid (VOF)** method. The two phases share one computational mesh and are identified by their volume fractions. For phase 1, the volume fraction is denoted by a_1, where

$$0 \leq a_1 \leq 1. \tag{51}$$

A value of $a_1 = 1$ represents a cell filled entirely with phase 1, while $a_1 = 0$ represents a cell filled entirely with phase 2. Values between zero and one indicate that the interface passes through the cell. The volume fraction of phase 2 is complementary:

$$a_2 = 1 - a_1. \tag{52}$$

The local mixture properties are obtained from the phase fractions. For example, the mixture density is represented by

$$\rho = a_1\rho_1 + a_2\rho_2, \tag{53}$$

where ρ_1 and ρ_2 are the densities of the two phases. The corresponding transport properties are determined from the phase composition and the material properties assigned to each phase. The interface is transported with the flow by solving a volume-fraction transport equation. Interface compression and boundedness controls are used to keep the volume fraction within its physical range and to maintain a reasonably sharp phase boundary. The resolved interface can be visualized using the $a_1 = 0.5$ isosurface.

The implemented model supports:

- two immiscible Newtonian fluids in one fluid domain;
- incompressible, isothermal flow;
- laminar flow and the $k - \varepsilon$ and $k - \omega$ SST turbulence models;
- steady calculations using local time stepping;
- transient calculations with adaptive time-step control;
- gravity and independently specified gravitational acceleration;
- surface tension between the two phases;
- phase-fraction boundary conditions at inlets and pressure or open outlets;
- wall adhesion through a constant static contact angle;
- thin walls with either regular no-slip or frictionless slip behavior.

The surface tension coefficient is specified for the pair of fluids. Gravity may be enabled or disabled independently of the flow solution. When gravity is active, the acceleration components define the direction and magnitude of the body force acting on the mixture.
At inlets, the phase-1 volume fraction defines the composition of the incoming fluid. At pressure or open outlets, the phase-1 fraction defines the composition used during backflow. The phase-2 fraction is always obtained as the complementary value. The initial phase distribution is assigned to the computational cells before the simulation begins. CFxD supports uniform filling, level-based filling, and box-based filling. For level and box definitions, the initial phase is selected according to the position of the cell center. Consequently, the discrete shape of the initial interface depends on the mesh resolution.

At solid walls, the phase fraction may use a zero-gradient condition or a constant static contact angle. The contact angle is measured through phase 1 and is applied to the wall–fluid interface. Dynamic contact-angle models are not currently exposed. For VOF thin-wall interfaces, the same adhesion setting is applied on both sides of the wall.

Steady VOF calculations use local time stepping to seek stationary flow and interface. The iteration count represents the computational stopping limit, while the local time step is a pseudo-time quantity rather than physical elapsed time. Steady calculations are therefore appropriate only when the flow and interface are expected to approach a stationary configuration.

Transient calculations use physical time advancement and are required for waves, sloshing, filling, draining, interface oscillations, and other time-dependent phenomena. The time-step control accounts for both the flow Courant number and the phase-fraction Courant number. The entered time step acts as an upper limit when adaptive stepping is enabled.

The VOF implementation is restricted to incompressible isothermal two-phase flow. Conjugate heat transfer, heat transfer, radiation, species mixing, combustion, Proudman acoustics, porous zones, rotating zones, fan interfaces, and porous interfaces are not supported within the same VOF case.

1.10 Combustion and reacting flow

Flows with combustion are important in engineering because combustion is widely used for energy generation, propulsion, heating, and industrial processing. Combustion is a chemical reaction in which fuel and oxidizer are converted into reaction products, usually with the release of heat. At the macroscopic level, combustion is governed by the interaction of convection, diffusion, heat transfer, and chemical reactions. The heat released by combustion changes the temperature, density, pressure, and composition of the flow. These changes influence the velocity field and may also modify the turbulence structure. Combustion modeling therefore requires the coupled solution of the flow, energy, and species-transport equations together with a model for the chemical reaction rate.

Chemical kinetics

Chemical kinetics, as an element of combustion modeling, describes the rates at which chemical reactions occur. In a kinetic description, the chemical reactions are considered through their reaction rates, while the effects of convection, diffusion, and heat transfer are represented separately in the governing transport equations. The conversion of reacting materials into combustion products can be written as a one-step reaction:

$$\nu'_F F + \nu'_O O = \nu''_P P, \tag{54}$$

where F is the number of fuel moles, O is the oxidizer, and P is the products. For example, a global methane–air reaction can be written as

$$\mathrm{CH_4} + 2\mathrm{O_2} \rightarrow \mathrm{CO_2} + 2\mathrm{H_2O}. \tag{55}$$

In reality, the oxidation of a fuel is usually more complex and may involve hundreds or thousands of elementary reactions and intermediate species. Taking into account an actual chemical system for N components and M elementary chemical reactions, it is possible to write the following system of equations:

$$\sum_{k=1}^{N} \nu'_{kj} \Psi_k = \sum_{k=1}^{N} \nu''_{kj} \Psi_k, j = 1, M, \tag{56}$$

where Ψ_k is the symbol to denote the components k and ν'_{kj} and ν''_{kj} is the molar stoichiometric coefficient of component k in reaction j. The system (56) should satisfy a mass conservation equation:

$$\sum_{k=1}^{N} \nu_{kj} W_k = 0, j = 1, M, \tag{57}$$

where $\nu_{kj} = \nu''_{kj} - \nu'_{kj}$ and W_k denotes the molecular weight of component k. The rate $\dot{\omega}_{kj}$ of forming component k in the reaction j is written as

$$\dot{\omega}_{kj} = r_j W_k \nu_{kj}, \tag{58}$$

where r_j is the rate of reaction j. The total rate $\dot{\omega}_k$ forming component k is written as the summary of all rates M of elementary reactions:

$$\dot{\omega}_k = \sum_{j=1}^{M} \dot{\omega}_{kj} = W_k \sum_{j=1}^{m} r_j \nu_{kj}. \tag{59}$$

From (56) and (59) it is possible to write:

$$\sum_{j=1}^{N} \dot{\omega}_k = \sum_{j=1}^{M} \left(r_j \sum_{k=1}^{N} W_k \nu_{kj} \right). \tag{60}$$

The rate of reaction, r_j, can be found as the difference between the direct and reverse reaction rates:

$$r_j = r_{f,j} - r_{b,j}, \tag{61}$$

where $r_{f,j} = K_{f,j} \Pi_{k=1}^{N} \left(\frac{\rho Y_k}{W_k} \right)^{\nu'_{kj}}$ and $r_{b,j} = K_{b,j} \Pi_{k=1}^{N} \left(\frac{\rho Y_k}{W_k} \right)^{\nu''_{kj}}$. The *rate of direct reaction*, $K_{f,j}$, can be determined by applying the *Arrhenius law*:

$$K_{f,j} = A_{f,j} T^{n_{f,j}} \, exp\left(-\frac{E_{f,j}}{RT} \right), \tag{62}$$

that has three empirical coefficients: a pre-exponential coefficient, $A_{f,j}$, a temperature coefficient, $T^{n_{f,j}}$, and an activation energy, $E_{f,j}$. The rate of reverse reaction, $K_{b,j}$, can be determined from the rate of direct reaction and equilibrium constant:

$$K_{b,j} = K_{f,j} \left[\left(\frac{p_a}{RT} \right)^{\sum_{k=1}^{N} \nu_{kj}} exp\left(\frac{\Delta S_j^0}{R} - \frac{\Delta H_j^0}{RT} \right) \right]^{-1}, \tag{63}$$

where p_a is the ambient pressure and ΔS_j^0, ΔH_j^0 are entropy and enthalpy change during a reaction. The main task of chemical-kinetics modeling is to identify the relevant species and reactions and to determine the corresponding kinetic and thermodynamic coefficients.

Turbulent combustion

Most energy-generating technical devices operate with turbulent combustion. In general, we will describe the turbulent combustion modeling in the context of RANS here. In turbulent combustion, significant fluctuations of density appear. In this case, the averaged equations will have additional correlations, including density fluctuations. To avoid it, an *averaging* suggested *by Favre* in 1969 is used. The averaging for variable f is written as

$$\tilde{f} = \frac{\overline{\rho f}}{\bar{\rho}}, \tag{64}$$

where the overbar denotes Reynolds averaging. For velocity, the Favre average is

$$u = \tilde{u}_i + u_i'', \tag{65}$$

while for scalar, it is

$$\varphi = \tilde{\varphi}_i + \varphi_i'', \tag{66}$$

where $\tilde{u}_i$, $\tilde{\varphi}$ are the mean parts and u'', φ'' are fluctuating parts.

The Favre-averaged conservative laws can be re-written as

$$\frac{\partial \overline{\rho}}{\partial t} + \frac{\partial \bar{\rho}\tilde{u}_i}{\partial x_i} = 0, \tag{67}$$

$$\frac{\partial \bar{\rho}\tilde{u}_i}{\partial t} + \frac{\partial \overline{\rho}\tilde{u}_j\tilde{u}_i}{\partial x_j} = -\frac{\partial \overline{p}}{\partial x_i} - \frac{\partial}{\partial x_j}\left(\bar{\rho}\nu\frac{\partial \tilde{u}_i}{\partial x_j} + \bar{\rho}\widetilde{u_i''u''}_j\right) + \bar{\rho}g_i, \tag{68}$$

$$\rho\frac{\partial \overline{\rho}\tilde{\varphi}}{\partial t} + \frac{\partial \overline{\rho}\tilde{u}_j\tilde{\varphi}}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\rho D\frac{\partial \overline{\varphi}}{\partial x_j} - \bar{\rho}\widetilde{u_j''\varphi''}\right) + \overline{\dot{\omega}_\varphi}. \tag{69}$$

where $\overline{\dot{\omega}_\varphi}$ is the chemical source term. Besides the closing problem of the above equations, at the combustion modeling, an important role is played by the *interaction* between the *turbulent and chemical components* of combustion, which is called *turbulence-chemistry interaction*. In addition to solving the averaged transport equations, it is necessary to model the mean chemical source term $\overline{\dot{\omega}_\varphi}$ produced by the interaction between turbulence and chemical reactions. Because the reaction-rate expression is nonlinear in temperature and species concentration, the mean chemical source term cannot generally be evaluated by substituting averaged quantities into the instantaneous reaction-rate equation, the value of the chemical source term cannot be written as

$$\overline{\dot{\omega}_\varphi} \neq \dot{\omega}(\bar{T}, \bar{Y}_\alpha), \tag{70}$$

and similarly for density

$$\overline{\rho} \neq \rho(\bar{T}, \bar{Y}_\alpha). \tag{71}$$

This closure problem is referred to as turbulence-chemistry interaction. The combustion model must provide an approximation for the mean chemical source term.

Transport equation for species

The transport equation for each species i' is written as

$$\frac{\partial}{\partial t}(\rho m_{i'}) + \frac{\partial}{\partial x_i}(\rho u_i m_{i'}) = \frac{\partial}{\partial x_i} J_{i',i} + R_{i'} + S_{i'}, \tag{72}$$

where $J_{i',i}$ is the diffusive flux; $R_{i'}$ is a *rate of formation* or *decomposition of specie*; and S_i is the source term. The diffusive flux $J_{i',i}$ is usually modeled by using a gradient assumption similar to the scalar transport equation, e.g.,

$$J_{i',i} = -\left(\rho D_{i',m} + \frac{\mu_t}{Sc}\right)\frac{\partial m}{\partial x_i}. \tag{73}$$

The transport equation (72) for species needs to be solved for $N-1$ species, where N is the total number of chemical components in the system. Some common methods to determine the rate of formation or decomposition of specie R_i are presented below.

The simplest way to define R_i is to neglect the turbulent fluctuations, i.e., to use *Arrhenius equation* (62), which is correct for a laminar combustion. However, such a method does not give precise results for turbulent combustion due to the non-linearity of equation (70) but can be used for the combustion with the small turbulent fluctuations. More precise and commonly used is the *model* suggested by Magnussen and Hjertager, which is based on the *eddy dissipation approach*. The eddy dissipation model connects a reaction rate and an eddy dissipation rate of reagents and products of combustion. In this model, the reaction rate $R_{i',k}$ can be found from the limiting value of two equations:

$$R_{i',k} = \nu'_{i',k} M_i A \rho \frac{\varepsilon}{k} \frac{m_r}{\nu'_{r',k} M_r}, \tag{74}$$

$$R_{i',k} = \nu'_{i',k} M_i A B \rho \frac{\varepsilon}{k} \frac{\sum_r m_r}{\sum_j^N \nu'_{r',k} M_r}, \tag{75}$$

where m_r represents the mass fraction of each specie; m_r is the mass fraction of reagents; and $A = 4.0$ and $B = 0.5$ are the empirical constants.

CFxD currently provides only a methane-air combustion model based on the global reaction (55). The supported species are methane CH_4, oxygen O_2, carbon dioxide CO_2, water vapor H_2O, and nitrogen N_2. Nitrogen is treated as an inert species. For laminar combustion, CFxD uses finite-rate Arrhenius chemistry. For turbulent combustion, CFxD uses the Eddy Dissipation Model, in which the reaction rate is controlled primarily by turbulent mixing. The model uses the turbulence dissipation rate and a cell-scale mixing contribution to estimate the local mixing rate. Combustion modeling requires a Pro license.

The reacting mixture is treated as an ideal gas with constant heat-capacity and transport-property assumptions. Species properties and formation enthalpies are obtained from the material definitions used by CFxD. Chemical heat release is coupled with the energy equation, and the resulting temperature and density changes affect the flow solution. The current implementation provides a foundation for future extensions of the combustion model. Additional fuels, reaction mechanisms, intermediate species, detailed finite-rate chemistry, pollutant formation, alternative turbulence-chemistry interaction models, premixed combustion, partially premixed combustion, ignition models, and multiphase combustion may be added within the same general reacting-flow framework.

At present, user-defined reaction mechanisms, detailed multi-step chemistry, spray combustion, solid-fuel combustion, and active continuous ignition-source modeling are not currently available.

1.11 Porous resistance models

Porous resistance models represent thin or volumetric structures whose detailed geometry would be impractical to resolve explicitly with the mesh. Typical examples include screens, meshes, perforated plates, filters, heat-exchanger cores, and other flow restrictions. Instead of modeling each opening or element, the porous model introduces an additional momentum loss that reproduces the dominant pressure-drop behavior while keeping the mesh size and computational cost manageable.

CFxD represents porous media in two ways. A **porous jump** is used for a thin resistive structure when most of the pressure loss occurs across an internal surface. A **porous zone** is used for a thick or distributed medium, such as a filter, packed region, or porous block, when the momentum loss must be applied throughout a volume.

The **Simple** model uses a dimensionless loss coefficient. Its pressure-drop relation is

$$\Delta p = \xi_i \frac{\rho u_i^2}{2} L, \tag{76}$$

where ξ_i is a directional loss coefficient per unit length, u_i is the velocity component in direction i, ρ is the fluid density, and L is the porous-media thickness. For a porous jump interface CFxD uses $L = 1\ m$ as a normalized thickness. For a porous zone, L represents the reference length over which the resistance is distributed through the cell zone. In a porous zone, separate simple-model coefficients may be specified along the global X, Y, and Z directions.

CFxD also supports a **Viscous/Inertial** model, which separates the pressure loss into linear and quadratic contributions:

$$\Delta p = (A_i u_i + B_i |u_i| u_i) L, \tag{77}$$

where A_i is the viscous resistance coefficient and B_i is the inertial resistance coefficient in direction i. The linear term represents losses that vary approximately in proportion to velocity, while the quadratic term represents inertial losses that increase with the square of velocity. For a porous zone, these coefficients can be defined independently along the global X, Y, and Z axes, allowing anisotropic resistance to be represented. For a porous jump, the corresponding coefficients describe the resistance applied across the normal interface.

The two formulations serve different purposes. The Simple model is appropriate when the porous medium is characterized by an overall dimensionless pressure-loss coefficient. The Viscous/Inertial model is appropriate when separate linear and quadratic resistance data are available, or when the pressure loss must be represented over a wider range of flow velocities. A porous jump is intended for thin localized structures, whereas a porous zone is intended for resistance distributed throughout a finite volume. Coefficients must be finite and nonnegative and should correspond to the fluid properties, flow direction, and operating range of the modeled application.

1.12 Rotation modeling

Rotating-flow problems are common in engineering applications such as fans, impellers, mixers, and rotating machinery. CFxD provides two steady modeling approaches: a simplified fan interface and the **Multiple Reference Frame (MRF)** method.

The simplified **fan model** represents a fan as an internal interface with a prescribed pressure rise. It does not resolve the fan blades or physically rotate the mesh. Instead, the pressure increase is applied across the interface to approximate the overall performance of the fan. The flow direction is defined relative to the interface orientation. A prescribed swirl angle may also be used to represent the tangential velocity component produced by the fan. When swirl is included, hub and tip diameters define the annular region through which the rotating flow is applied. This model is suitable when the overall pressure-boosting effect is more important than the detailed flow field around the blades.

The **Multiple Reference Frame (MRF)** method provides a more detailed approximation for rotating-flow applications. It models a rotating region using a steady rotating reference frame. The mesh remains stationary, but the momentum equation inside the specified rotating zone is modified by adding rotation-related source terms. In the rotating frame, the apparent forces include **Coriolis** and **centrifugal** contributions, which account for the effect of rotation without physically moving the mesh.

MRF is applied to a rotating volumetric zone by specifying its rotation axis and angular velocity. Outside the rotating zone, the flow is solved in a stationary reference frame. The method is well suited for quasi-steady rotor–stator problems and provides an approximation of the mean flow field around rotating components.

Neither the simplified fan model nor MRF resolves the full time-dependent motion of the rotor. Effects such as blade passing, transient rotor–stator interaction, and rotating wakes require a transient sliding-mesh approach. The simplified fan model is intended for compact performance estimates, while MRF is appropriate when the volumetric influence of rotation on the surrounding flow must also be represented.

1.13 Mesh considerations

At the core of CFD are the **conservation laws** of **mass**, **momentum**, and **energy**, which must be solved across a spatial domain. To accomplish this, the domain is divided into small subregions — called **cells** or **elements** — through a process known as **meshing**.
The complete set of these cells forms the **computational mesh** (also referred to as a **grid**), which provides the geometric framework for solving the discretized equations.
In CFxD, which supports three-dimensional simulations, **hexahedral cells** are primarily used. Hexahedral meshes offer good numerical stability and performance, especially for structured and semi-structured domains.
The **quality** and **resolution** of the mesh significantly affect the accuracy, stability, and computational efficiency of a CFD simulation. Key impacts include:

- **Accuracy**: A finer mesh captures geometric details and sharp gradients more accurately, especially near walls, interfaces, or shock layers.
- **Stability**: Poor mesh quality (e.g., skewed or highly stretched elements) can cause solver divergence or numerical oscillations.
- **Performance**: Smaller cells increase computational costs. A balanced mesh strategy — fine where needed, coarse elsewhere — helps reduce runtime without sacrificing accuracy.

A poorly constructed mesh can lead to **unphysical results**, **non-converging solutions**, or **excessive computation times**.

1.14 Boundary conditions

In CFD, **boundary conditions (BCs)** define the behavior of the fluid at the edges of the computational domain. They are a critical part of the simulation setup, as they influence how fluid enters, exits, or interacts with surfaces in the domain.
CFxD allows users to define boundary conditions for:

- **Velocity**
- **Pressure**
- **Temperature**
- **Turbulence variables** (e.g., k, eps, ω)

Boundary Types in CFxD

Inlet

The **Inlet** boundary condition is used to specify the incoming flow entering the computational domain. CFxD provides the following inlet options:

Velocity Inlet

The flow velocity is prescribed directly using its components (**Dirichlet condition**).

- The user specifies the velocity vector.
- Turbulence quantities (e.g., k, ω, or ε) can be defined in the turbulence subpanel.
- Temperature can be specified when heat transfer is enabled.

This option is typically used when the inflow velocity profile is known.

Mass Flow Inlet

Instead of velocity components, the user specifies the **mass flow rate**.

- CFxD internally computes the corresponding velocity based on fluid properties and inlet area.
- Temperature can be prescribed at the inlet.
- Turbulence quantities can be specified when a turbulence model is active.

This condition is appropriate when the total mass flow entering the domain is known.

Freestream Inlet

The **Freestream** condition is intended for external flow simulations (e.g., flow around bodies).

- The user specifies freestream velocity and other flow properties.
- The boundary allows a natural adjustment of flow variables consistent with far-field conditions.
- It is commonly used for aerodynamic simulations where the inlet represents undisturbed ambient flow.

For compressible subsonic flow (M < 1), freestream conditions are interpreted accordingly using the specified velocity and thermodynamic properties.

Pressure Inlet

The pressure inlet condition is used only in combination with a **Mass flow** Outlet boundary condition.

- The inlet pressure is specified by the user.

Outlet

CFxD provides several outlet boundary conditions.

Pressure Outlet

This option imposes a fixed static pressure at the outlet boundary.

- In the user interface, the specified pressure value is treated as **gauge pressure** for both incompressible and compressible simulations.
- For **compressible subsonic flow (M < 1)**, CFxD internally converts the gauge value to **absolute pressure** using the reference (ambient) pressure defined in the Operating conditions.
- Velocity is typically assigned to a **zero-gradient** condition.
- Temperature is treated according to the selected thermal model and boundary configuration.

This condition is appropriate when the downstream pressure is known (e.g., discharge into the atmosphere or into a chamber with controlled pressure).

Open Outlet

The **Open** outlet is designed to allow flow to exit the domain with minimal artificial constraint.

- Pressure is defined by the user as a gauge value (converted internally to absolute pressure in compressible simulations).
- Velocity and other transported variables use an open condition, allowing reverse flow if it occurs. The user may specify a backflow velocity value, which is applied when the flow reverses at the outlet boundary.
- For temperature, the user may specify an outlet value that is applied in case of backflow, consistent with the open boundary condition.

This condition is useful when the exact outlet pressure is not strictly prescribed and a more flexible outflow treatment is required.

Mass Flow Outlet

The **Mass Flow** Outlet is used only in combination with Pressure Inlet boundary condition. It represents the target mass flow rate at the outlet.

- The target outlet mass flow rate at the outlet is specified by the user.

Wall

Walls represent **solid boundaries** in the computational domain. A typical wall condition includes:

- **No-slip velocity** condition: fluid velocity at the wall is set to zero
- **Thermal conditions**, which may include:
  - **Fixed temperature** (*Dirichlet condition*): the temperature at the wall is specified
  - **Zero-gradient** (*adiabatic or insulated*): no heat flux across the wall.

Symmetry

Symmetry conditions are used for idealized or simplified boundaries with no normal flux or no shears.

Interface

The interface represents an internal surface through which flow resistance, pressure rise, wall behavior, or thermal interaction is defined. CFxD supports four interface functions: thin wall, porous resistance, fan, and conjugate heat transfer.

A **thin wall** represents an internal wall with negligible or unresolved physical thickness. It may impose either a regular no-slip condition or a frictionless condition. When heat transfer is enabled, the thin wall may also define perfect thermal contact or a specified thermal resistance.

A **porous interface** represents a localized pressure loss caused by a thin resistive structure, such as a screen, mesh, perforated plate, filter layer, or baffle. The pressure drop is applied across the surface rather than distributed throughout a volume. CFxD supports a simple dimensionless loss coefficient and a viscous/inertial formulation using linear and quadratic resistance terms. These parameters describe the hydraulic resistance in the direction normal to the interface.

A **fan interface** represents a pressure-generating device across an internal surface. Its primary parameter is the pressure rise, with the flow direction defined relative to the interface orientation. The model can also represent swirl through a specified outlet angle and, where required, hub and tip diameters that define the annular fan region.

For **conjugate heat transfer (CHT)**, the interface couples adjacent fluid and solid regions through their shared internal surface. It enables heat transfer between the regions while preserving their separate thermophysical properties. The thermal connection may be modeled as perfect contact, which assumes no additional contact resistance, or as a resistive contact with a specified thermal resistance.

An interface is an internal boundary and must be associated with adjacent computational regions. It is distinct from external inlets, outlets, walls, and symmetry boundaries. The accuracy of the resulting pressure, velocity, and temperature fields depends on consistent interface geometry, correct pairing of neighboring regions, and adequate mesh resolution near the shared surface.

1.15 Limitations

CFxD is designed for general-purpose CFD simulations under common engineering conditions. However, it has certain limitations that described below.

Continuum Flow Assumption

CFxD is based on **continuum mechanics**, which assumes that fluid properties vary smoothly and continuously in space. This assumption breaks down when the fluid is too rarefied (e.g., at very low pressures or in micro/nanoscale geometries). To verify that the continuum assumption is valid, the **Knudsen number** is used:

$$Kn = \frac{l}{L}, \tag{78}$$

where l is a free path (average distance a molecule travels before colliding) and L is a characteristic length scale. The flow is considered **continuum** when the *Knudsen number* is much smaller than 1: $Kn \ll 1$. In such cases, the Navier–Stokes equations used in CFxD remain valid. For larger Knudsen numbers (Kn≳0.1), rarefied gas effects become significant, and CFD models based on continuum mechanics may no longer provide accurate results.

Newtonian Fluids

CFxD supports only the simulation of **Newtonian fluids**, where viscosity remains constant regardless of the flow conditions. Non-Newtonian effects such as shear-thinning, viscoelasticity, or time-dependent viscosity are not currently supported.

Single-Phase Flows

Only **single-phase** simulations are supported. CFxD does not currently handle multiphase models such as:

- Gas–liquid mixtures,
- Particle-laden flows (DPM),
- Free-surface flows (VOF),
- Cavitation or phase change phenomena.

No Chemical Reactions

CFxD does **not support reacting flows**. This includes combustion, oxidation, and species transport with source terms or reaction kinetics. All scalar fields (e.g., temperature) are treated as passive or purely diffusive.

The capabilities of CFxD are summarized in Table 1.

Table 1: CFxD capabilities.

Feature Category	Supported in CFxD
Mode	Steady, transient
Phases	Single and multiphase (VOF) flows, gas mixture
Fluid Type	Newtonian and non-Newtonian fluids
Compressibility	Incompressible, compressible (subsonic M<1, supersonic M≥1)
Flow Regimes	Laminar, turbulent (k–ε, SST)
Heat Transfer	Conduction, natural/forced convection, CHT, surface-to-surface
Chemical Reactions	Combustion (methane-air)
Acoustic	Empirical (Proudman model)
Porosity	Porous jump/Porous zone
Rotation	Rotating domain
Geometry Dimensions	3D
Meshing	Hexahedral/hex-dominant
Solver Interface	OpenFOAM integration
Post-Processing	3D visualization, probes, charts, contours, streamlines
Material Models	Constant properties, ideal gases

Chapter II. Getting started

2.1 Installation

CFxD is distributed as a standalone installer for supported operating systems. The **demo version** can be downloaded from the official CFxD website: https://cfxd.com/download/CFxD-win.exe.

Windows Installation

1. Download the installer file CFxD-win.exe from the link above.
2. Double-click the installer to start the installation process.
3. Follow the on-screen instructions of the installation wizard:
    - Choose the installation directory (recommended to use a default installation folder: C:\Program Files\CFxD)
    - Select the components (CFxD Application: core program, CFxD Runtime: required libraries, such as OCCT, OpenFOAM, VTK, Microsoft MPI: required for a multi-core simulation)
    - Accept the license agreement
4. Click **Install** and wait until the installation process is complete. If prompted by Windows security, confirm that you want to allow the application to make changes to your system.
5. Once finished, CFxD can be launched from:
    - the **Start Menu**, or
    - the **desktop shortcut** (if selected during installation).

First launch

On the first launch:

- CFxD initializes its internal folders and settings.
- The software starts in **Demo mode**, which includes a mesh-size limitation of 250000 cells and single CPU core usage.
- An active internet connection is required to check for updates and license status.

To remove the demo limitations, a valid license must be activated through the application interface.

System Requirements (Minimum)

- **Operating System**: Windows 10/11 (64-bit)
- **Processor**: Multi-core CPUs recommended
- **Memory**: 8 GB RAM (16 GB or more recommended for large meshes)
- **Disk Space**: At least 2 GB free space
- **Graphics**: OpenGL-compatible GPU

Actual hardware requirements depend on geometry complexity and mesh density.

Troubleshooting

- If the installer does not start, ensure you have administrator privileges.
- Temporarily disable antivirus software if installation is blocked.
- Make sure all required system updates for Windows are installed.

2.2 Graphical User Interface (GUI)

CFxD includes all essential elements for flow simulation in engineering applications, such as geometry creation and import/export, mesh generation, boundary condition setup, solution computation, and visualization. As an example, the workspace for geometry creation is shown in Figure 5.

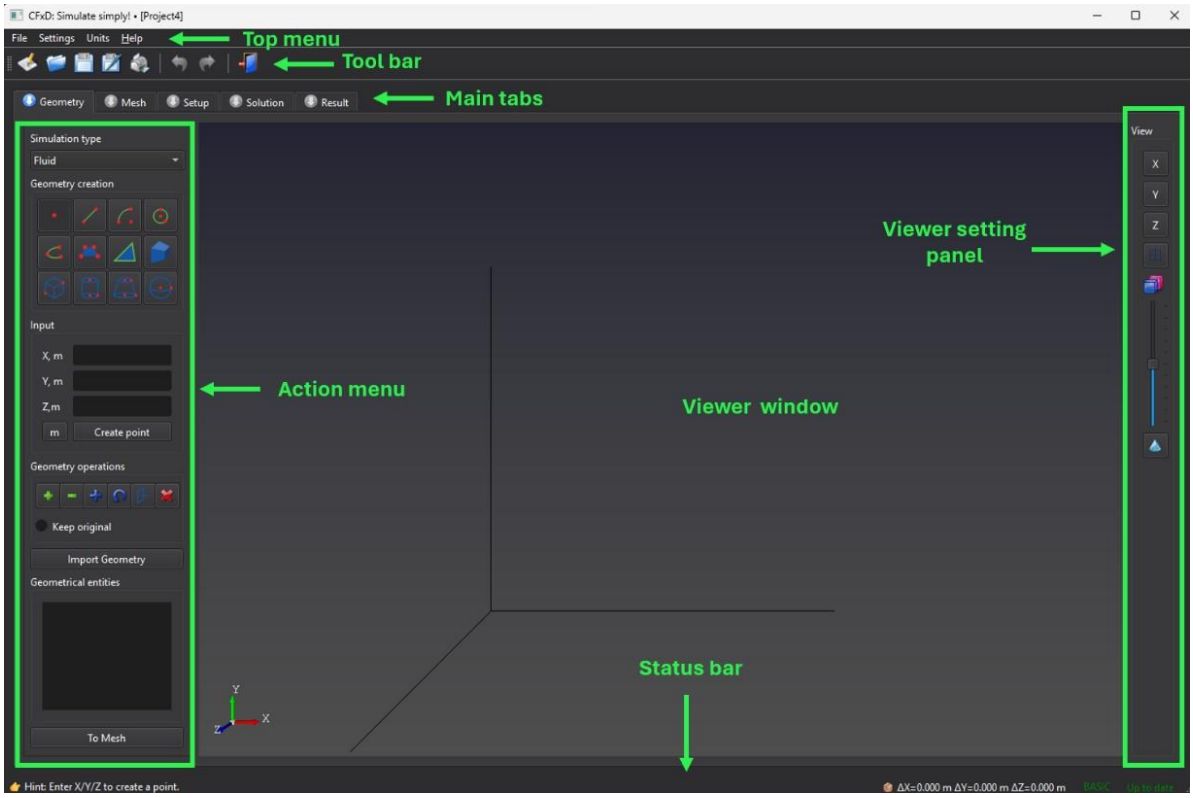


Figure 5. Working window for geometry creation.

The **Top menu** provides access to common operations, including file management, units change, report printing, help, and general program information. In addition, the Settings → Appearance menu allows switching between two interface styles: Classic and Modern (Figure 6).

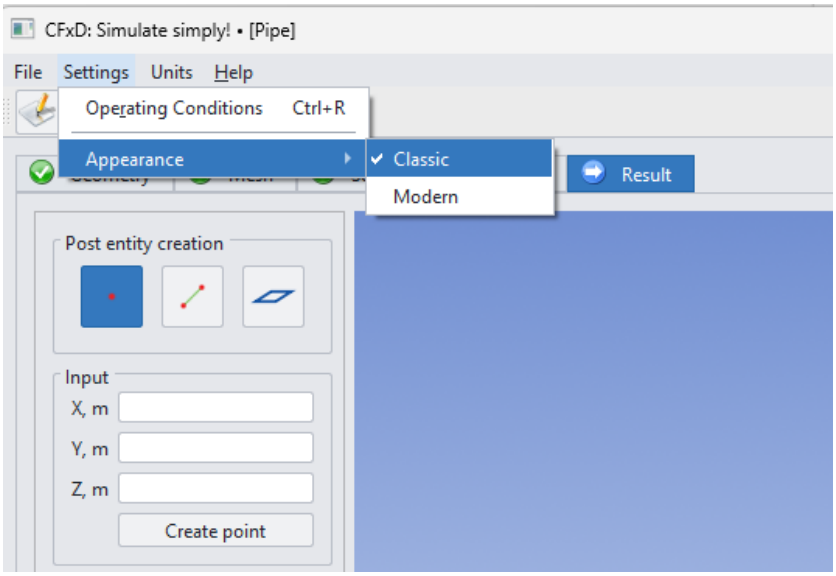


Figure 6. Appearance menu.

The Toolbar, located below the top menu, provides quick access to New, Open, Save, Save As, Print and Exit. On the Geometry tab it also displays Undo and Redo. These actions operate on recorded geometry creation steps and are enabled only when the corresponding history is available; they do not provide a general undo history for mesh, setup or solver changes.

Below the toolbar are the **Main tabs** (Figure 7) required for flow simulation: **Geometry, Mesh, Setup, Solution, and Result**. The icon indicates completed tabs. The icon indicates incomplete tabs.



Figure 7. Main tabs.

Each tab is divided into two main sections: a vertical **Action menu** on the left and **Viewer window** for visualization on the right (Figure 5). The tabs with **Viewer window** additionally have the **View setting panel** on the right side.

The Status bar is located at the bottom of the window. It displays hints for the active operation, overall geometry dimensions, the license type and update status. The dimensions are the bounding-box extents in the X, Y and Z directions. An internet connection is required to check for updates. When an update is available, the update button opens release information and asks for confirmation before closing CFxD and starting the updater.

The general workflow starts with the vertical action menu, following a top-to-bottom sequence, and then progresses horizontally to the next tab. A detailed description of each tab and buttons is provided in the following chapters.

Project commands

Use File > New to start a project or File > Open to load a .cfxd project. The window title identifies the current project. Save records the current data; Save As saves under another name or location. When copying a case, keep its project file and associated folders together.

Save regularly and before changing projects or closing CFxD. Generate simulation reports from Result. File > Print offers the log and, when a report exists, Report (PDF). Selecting the PDF opens it for viewing or printing.

Common keyboard shortcuts are Ctrl+N for New, Ctrl+O for Open, Ctrl+S for Save and Ctrl+P for Print. Ctrl+R opens Operating Conditions, Ctrl+L opens the Log Window and Ctrl+M opens the Manual. Press F1 for help on the control under the pointer.

Workflow tabs and stage transfers

A new project begins in Geometry. Later tabs become available as the required data is transferred. Selecting a tab changes the workspace; the transfer button prepares the next stage. Work down the left panel, inspect the viewer, then use To Mesh, To Setup, To Solution or To Result.

Geometry contains the Simulation type selector, creation and editing tools, coordinate inputs and the geometry tree. Create or import the volumes and name important boundary faces before using To Mesh. Mesh contains the base cell size, local refinement and zone settings, generation controls, statistics and quality inspection. Generate and inspect the mesh before using To Setup.

Setup contains the flow and thermal models, time or iteration settings, domain materials, boundary conditions and CPU-core setting. Review the Domains and Boundaries trees and apply the required values in their dialogs before using To Solution. Solution contains run controls, residual and monitoring graphs, balance indicators and solver output. Result contains the saved-field viewer, post-processing entities, display settings, screenshots and report tools.

The tab icons indicate workflow progress. They do not certify the quality of the physical model or numerical result. Inspect mesh quality and check residuals, monitored quantities and conservation balances separately. If geometry, mesh or setup inputs are changed, repeat the affected preparation and transfer steps before relying on a later solution.

Action panels and selection trees

The left action panel groups the controls for the active task. Some buttons select a tool, while others open a dialog or execute an operation. In Geometry, select the required creation or operation tool first, then select the defining entities in the order described in Chapter III. The active tool determines whether points, curves, faces or bodies can be selected.

Use selection trees to locate model entities. Expand a Geometrical entities category, click an entry to identify it in the viewer and use its checkbox to control visibility. In Setup, select a domain or boundary to edit its properties. In Result, select a Post entities entry to open its value, chart or display dialog.

A visibility checkbox hides or shows an object. Delete removes the geometry or post-processing definition. Right-click an entry for the commands available for that tab and object.

Viewer navigation and display controls

Use the viewer to inspect the model after each operation. Rotate the model by dragging with the middle mouse button, pan by dragging with the right mouse button, and zoom with the scroll wheel. In Geometry, a right-click without dragging can select a face for naming. Left-click selection follows the active tool. Detailed mouse commands are given in Section 3.1.

The view panel on the right provides axis views and Fit all. Use Fit all when an object is outside the current view or after changing the model extent. Additional controls depend on the tab: Geometry and Result include transparency and wireframe display, while Mesh includes section cuts and a coordinate scale. Display changes help inspect the model; they do not alter the geometry dimensions or mesh resolution.

Input values units and control availability

Before entering physical values, choose Units > SI: kg, m, s or Units > Imperial: lb, ft, s, and read the unit printed beside each field. The Geometry Input panel also has a length-unit button for m/mm or ft/in. This small-length selection applies to Geometry input; check the units in other dialogs separately. Operating Conditions labels its reference values in Pa, K and m/s^2.

Enter the required values and use the dialog action specified for that task, such as Apply, Add or Create. Preview is used where available to inspect a proposed entity or setting before adding it. Changing a selector can enable additional inputs, such as turbulence or thermal values, so review the newly available fields before continuing.

A disabled control can indicate missing prerequisite data or a restriction imposed by the active model, workflow stage or license. Check the preceding transfer and any application message. Settings > Appearance selects Classic or Modern. Settings > Operating Conditions opens the reference-pressure and buoyancy controls described in Section 5.2.

Hints messages and help

Read the status-bar hint for the active tool. Check the displayed X, Y and Z extents against the expected model size and units. The license indicator identifies the active license; application messages explain operation-specific restrictions.

For instructions on a particular control, place the pointer over it and press F1. If no specific topic is available, CFxD tries the focused control and then opens the help contents. Help > Manual opens the help browser. Within that window, browse Contents or press Ctrl+K to search article titles and text, then press Enter to open a matching topic. All entered search words must match, without regard to letter case.

For troubleshooting, open Help > Log Window and review the messages around the failed operation. During a calculation, also inspect the solver output in Solution. The log records application actions and messages, while the solver output provides details of the numerical run. Further descriptions of the history log and interactive help follow in Sections 2.4 and 2.5.

2.3 File structure

The project folder structure is presented in Figure 8. The projects are saved by default in **Documents** folder. The folder structure represents the workflow described previously. The geometrical entities are saved in the **Geometry** folder, mesh settings and mesh in the **Mesh** folder, the settings and boundary conditions in the **Setup** folder, solver files in the **Solution** folder and output files and report in the **Result** folder. The general data is saved in main file with .cfxd extension.

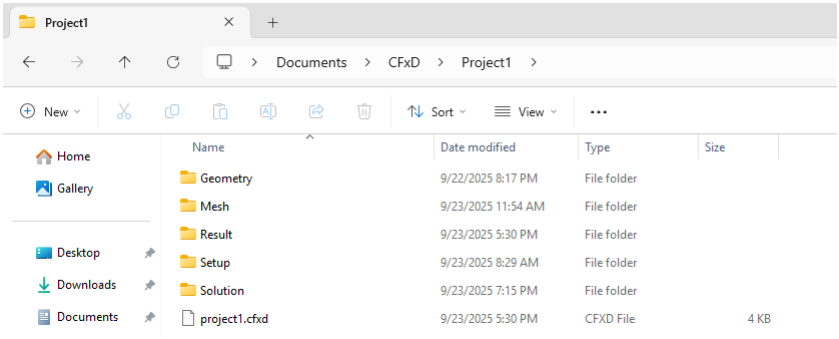


Figure 8. Folder structure.

The file structure is described in Table 2. Project file and main folder names are automatically generated with numbers: project1.cfxd in the folder Project1, project2.cfxd in the folder Project2, etc. The project file is written in JSON format and includes all information about files and user inputs.

Table 2: File structure.

Folder	Sub folder	Files (description)
		projectX.cfxd – main file
Geometry/		*.brep - native OCCT file *.stl - exported geometry
Mesh/	0/	default empty folder
	constant/	extendedFeatures – surface feature data edgeMesh – edges mesh data polyMesh – mesh file triSurface - surface geometry files
	system/	blockMeshDict – structured mesh control file snappyHexMeshDict – unstructured hexa-mesh control file surfaceFeatureDict – sharp edges detection control file
Setup/	0/	U – initial and boundary values of velocity T – initial and boundary values of temperature p/p_rgh – initial and boundary values of pressure k – initial and boundary values of turbulent kinetic energy omega – initial and boundary values of turbulence dissipation rate nut – turbulent kinematic viscosity, derived or explicitly initialized alphat – turbulent thermal diffusivity, derived or explicitly initialized
	constant/	transportProperties - transport properties turbulenceProperties - turbulence properties
	system/	controlDict – simulation control file decomposeParDict – multicore computing control file fvSchemes – discretization scheme settings file fvSolution – solution numerical settings file
Solution/	0/ X/ constant/ system/ postProcessing/	U, T, p, k, epsilon, omega, nut, etc. output statistics files files for post-processing files
Result/		images, export data report.pdf – report file with simulation results

2.4 History log

The history log window (Figure 9) can be opened from the top menu by choosing Help → Log Window. The history log window records important actions made by the user. The log is also useful for troubleshooting and debugging.

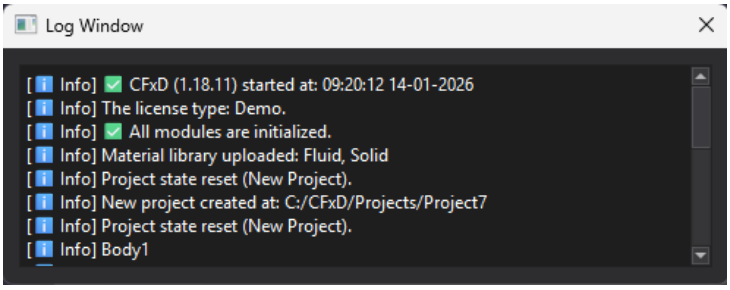


Figure 9. History log.

2.5 Interactive help

The interactive help can be opened from the top menu by choosing **Help → Manual**. The help window provides a structured description of how to use the program. Additionally, you can open context-sensitive help for the related topic by pointing the mouse cursor at an element and pressing **F1** (Figure 10).

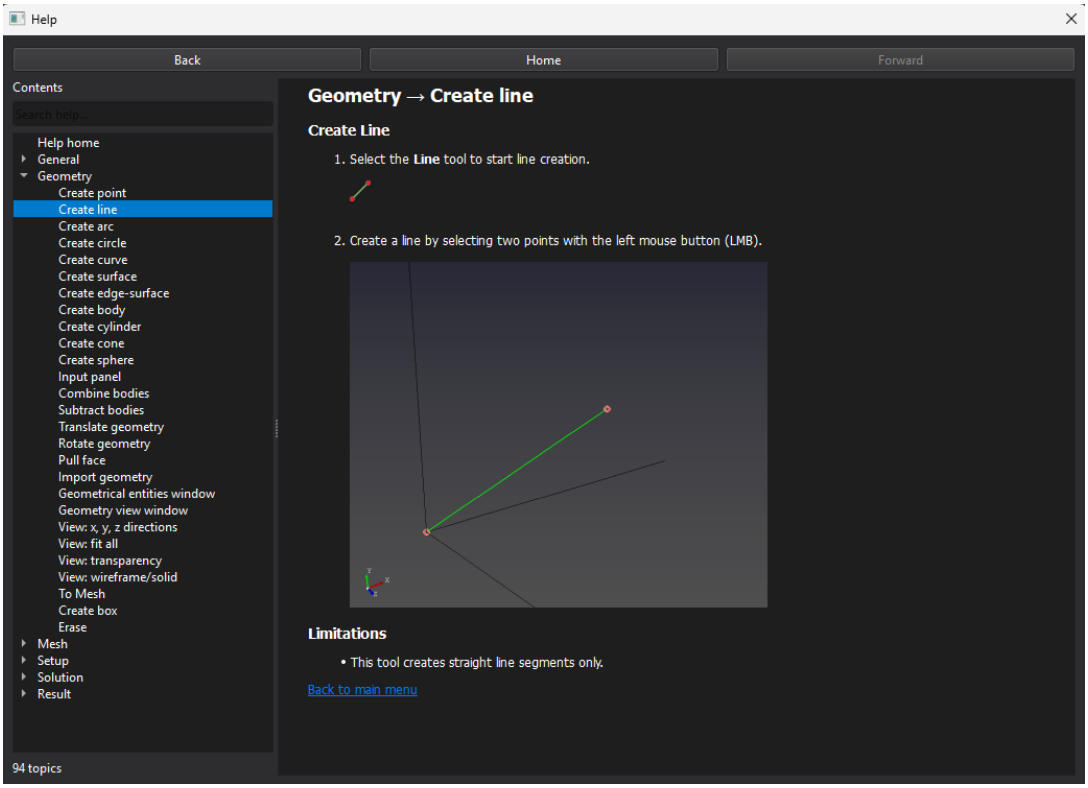


Figure 10. Interactive help opened with F1.

2.6 Licenses

CFxD is distributed with Demo, Basic, and Pro license tiers. The license tier determines the maximum case size, the maximum number of CPU cores, and access to selected advanced physics models. The main limits are summarized in Table 3.

Table 3: License type.

License type	Demo	Basic	Pro
Maximum number of cells	250,000	5 million	100 million
Maximum CPU cores	1	5	100
Advanced physics	No	No	Yes

The Demo license is intended for evaluation and small test cases. It limits the mesh size to 250,000 cells and the simulation to one CPU core. The Basic license supports cases with up to 5 million cells and five CPU cores. The Pro license supports cases with up to 100 million cells and 100 CPU cores and enables the advanced physics models available in CFxD.

CFxD provides extended physics models in addition to the basic single-phase Newtonian flow model. Basic licenses support conjugate heat transfer, porous jump and porous-zone resistance, and simplified fan-interface modeling.

The following models require a Pro license:

- non-Newtonian Power-law viscosity,
- MRF rotating zones,
- gas-mixture species transport,
- methane–air combustion,
- Volume-of-Fluid modeling for two immiscible fluids,
- Proudman broadband acoustics.

The availability of a model also depends on the simulation type and on compatibility with other enabled models. For example, combustion and gas-mixture modeling require a fluid simulation, VOF requires a fluid simulation with incompressible isothermal flow, and Proudman acoustics requires turbulent flow. VOF cannot be combined with heat transfer, combustion, acoustics, porous models, rotating zones, or fan interfaces.

Projects created with Pro-only physics may still be opened for inspection under a lower license tier, subject to the applicable limits. Case generation, solver execution, or modification of a Pro-only model requires a valid Pro license.

2.7 Typical workflows

A typical CFD simulation in CFxD consists of five steps: Geometry, Mesh, Setup, Solution and Result. The following sections summarize common workflows for different simulation types.

General flow

A general flow case is a **single-phase** simulation without heat transfer and without additional models such as porous resistance or rotating flow. The main steps are summarized in Table 4.

Table 4: General flow.

Geometry	• Create or import geometry in CFxD. • Ensure the flow domain is a single closed body. • Name boundary faces (e.g., inlet, outlet, wall, symmetry).
Mesh	• Generate the base mesh. • Apply local surface and/or volume refinement if required.
Setup	• Assign a material to the fluid domain. • Define boundary conditions and operating conditions.
Solution	• Run the solver and monitor residuals and key quantities.
Result	• Visualize results (contours, vectors, plots) and generate a report if needed.

Examples of general flows can be found in Tutorial 1 (incompressible internal flow), Tutorial 2 (incompressible external flow) and Tutorial 5 (low Mach compressible flow).

General flow with heat transfer

The general flow with heat transfer includes cases with wall heat transfer (fixed temperature or heat flux), thermal mixing (the same fluid at different inlet temperatures) and natural convection (buoyancy driven flow). The additional setup steps are summarized in Table 5.

Table 5: General flow with heat transfer.

Setup	• Assign a material with thermal properties to the fluid domain. • Enable the **Heat Model**. • Define boundary conditions with thermal inputs (e.g., inlet temperature, wall temperature/heat flux) and set operating conditions (reference pressure). • For natural convection, enable **gravity/buoyancy** in **Operating conditions**.

Examples of general flows can be found in Tutorial 3 (flow with heat transfer) and Tutorial 4 (natural convection).

Flow with internal wall (baffle)

This workflow applies to thin internal features (e.g., plates, screens, or baffles) that influence the flow but do not require modeling heat conduction within a solid region. The feature is represented as an **internal wall surface** inside the fluid domain. Thermal boundary conditions can be applied on this surface (e.g., adiabatic or fixed temperature) to model heat transfer between the fluid and the wall. The same internal surface can also be used to define a **porous jump**. The specific steps are summarized in Table 6.

Table 6: Flow with internal wall (baffle).

Geometry	• Create an internal surface. • Optionally, assign a name to the surface (interface).
Mesh	• Apply Surface Refinement around the internal surface if required.
Setup	• Assign the **Interface** as a **Wall** boundary, or as a **Porous** to model porous jump.

Flow with conjugate heat transfer (Pro license)

This workflow includes thin solid inserts inside a fluid domain (e.g., fins, thin plates, heat spreaders) where heat conduction through the solid is important. The model is configured as **conjugate heat transfer (CHT)** with separate **fluid** and **solid** domains coupled through an internal interface. This allows the solid temperature field and the heat flux through the solid thickness to be resolved. The specific steps are summarized in Table 7.

Table 7: Flow with conjugate heat transfer (CHT).

Geometry	• Select **Conjugated heat transfer** in **Simulation type**. • Create or import two bodies (fluid and solid). • Optionally, assign a name to the shared surface (interface).
Mesh	• Apply Surface Refinement around the shared surface if required.
Setup	• Assign a material to the fluid domain. • Assign a material to the solid domain. • Define the interface as Perfect contact or Resistance.

Flow with porous media

CFxD models porous resistance using either a **porous jump** (thin resistance across an internal interface) or a **porous zone** (volumetric resistance applied within a defined region). In both cases, resistance is specified using directional loss coefficients (X/Y/Z). CFxD supports both the **Simple** and **Viscous/Inertial** resistance formulations. The porous jump workflow is described in **Flow with internal wall (baffle)**. The specific steps for porous zone are summarized in Table 8.

Table 8: Flow with porous media.

Mesh	• Create a porous zone in **Refinement | Zones** dialog.
Setup	• Set the directional loss coefficients in the **Porous zone** dialog opened from **Domains tree** under **Porous**.

Flow with rotation (MRF with Pro license)

CFxD models rotating-flow applications using either a simplified fan interface or the Multiple Reference Frame (MRF) method. The specific steps for rotation modeling are summarized in Table 9.

Table 9: Flow with rotating frame.

Mesh	• Create a **Rotating zone (Cylinder)** in **Refinement | Zones** dialog.
Setup	• Set the rotation axis and rotational speed in the **Rotating zone** dialog opened from **Domains tree** under **Rotation**.

Flow with gas mixture (Pro license)

This workflow applies to mixing of two gases in a single fluid domain, with or without temperature differences between the inlet streams. In CFxD, gas-mixture cases are modeled as

compressible species-transport simulations, where the composition is defined at the inlets using mixture fractions. The specific steps are summarized in Table 10.

Table 10: Flow with gas mixture.

Setup	• Enable **Gas mixture**. • Set two gases in the **Domain properties** dialog opened from the **Domains tree** under **Fluid**. • Define mixture fractions of the two gases in the **Inlet** dialog opened from the **Boundaries tree** under **Inlet**.

Acoustics modeling (Pro license)

CFxD provides a simplified broadband acoustic model based on the Proudman analogy. The model estimates acoustic power generated by isotropic turbulence from the flow and turbulence fields. It requires turbulent flow, is available for fluid simulations, and can be used in steady or transient cases. The model is not available for VOF simulations. The additional setup and result steps are in Table 11.

Table 11: Acoustics modeling.

Setup	• Select turbulent flow and enable **Physics → Acoustics → Proudman**.
Result	• Display **Acoustic Power Density** P_A and **Acoustic Power Density Level** L_P. Use **Sound Power Report** to integrate the acoustic power over the fluid volume.

Proudman acoustics requires turbulent flow and a Pro license.

Multiphase flow modeling (Pro license)

CFxD models multiphase flow using the Volume-of-Fluid (VOF) method. The model is intended for two immiscible Newtonian fluids sharing one fluid domain, such as liquid-gas flow, filling, draining, sloshing, and interface motion. The additional setup and result steps are summarized in Table 12.

Table 12: Multiphase flow.

Setup	• Select the Fluid simulation type and enable **Physics → Multiphase → Volume of Fluid (VOF)**. Select both materials under Domains and specify the surface-tension coefficient. • Define gravity and the gravitational-acceleration components when buoyancy or free-surface effects are required. • Specify the initial phase distribution using a uniform region, level, or box. • Define the phase-1 volume fraction at inlets and backflow outlets. For walls, select the wall-adhesion option and, if required, specify the static contact angle.
Solution	• Select steady or transient operation. Transient calculations are recommended for waves, sloshing, filling, draining, and other time-dependent interface motion.
Result	• Display the phase volume fractions, mixture density, and the phase-1 volume-fraction isosurface at $\alpha_1 = 0.5$ to visualize the interface.

VOF requires the Fluid simulation type and a Pro license. It is restricted to incompressible, isothermal flow with two immiscible fluids. Heat transfer, CHT, species mixing, combustion, Proudman acoustics, porous interfaces, porous zones, rotating zones, and fan interfaces are not supported in the same VOF case.

Flow with combustion (Pro license)

CFxD models combustion as a reacting-flow simulation with coupled flow, energy, and species transport. The current implementation supports a methane–air reaction mechanism for fluid simulations. Laminar combustion uses finite-rate Arrhenius chemistry, while turbulent combustion uses the Eddy Dissipation Model (EDM). The additional setup and result steps are summarized in Table 13.

Table 13: Flow with combustion.

Setup	• Select the **Fluid** simulation type and enable **Physics → Species & Combustion → Combustion**. • Select the **Methane–Air** reaction mechanism and assign the required species materials under Domains. • Define the species composition at inlets and outlets using the available presets or user-defined mass fractions. • Select laminar or turbulent flow. Laminar cases use finite-rate Arrhenius chemistry; turbulent cases use the Eddy Dissipation Model. • Define the temperature, velocity, pressure, and turbulence boundary conditions required by the reacting-flow model.
Solution	• Select steady or transient operation and generate the reacting-flow case.
Result	• Display temperature, species mass fractions, density, velocity, pressure, and other available flow variables to evaluate the combustion process.

Combustion requires the Fluid simulation type and a Pro license and is not available with VOF or CHT. The current implementation supports only the methane–air mechanism and does not include arbitrary user-defined reaction mechanisms, detailed multi-step chemistry, spray combustion, solid-fuel combustion, or an active continuous ignition-source model.

Chapter III. Geometry

The **Geometry** tab includes a vertical **Action menu** with options for basic geometry creation, coordinate and movement input, geometry operations, and a **Geometry entities** tree displaying the created elements. Geometrical elements are visualized in the **Viewer window** (Figure 11). The **Viewer settings** panel is located on the right side of the **Viewer window**.

In addition to the geometry tools, the **Action menu** includes a **Simulation Type** selector at the top. It provides currently two options - **Fluid** and **Conjugate Heat Transfer** - which streamlines the subsequent workflow.

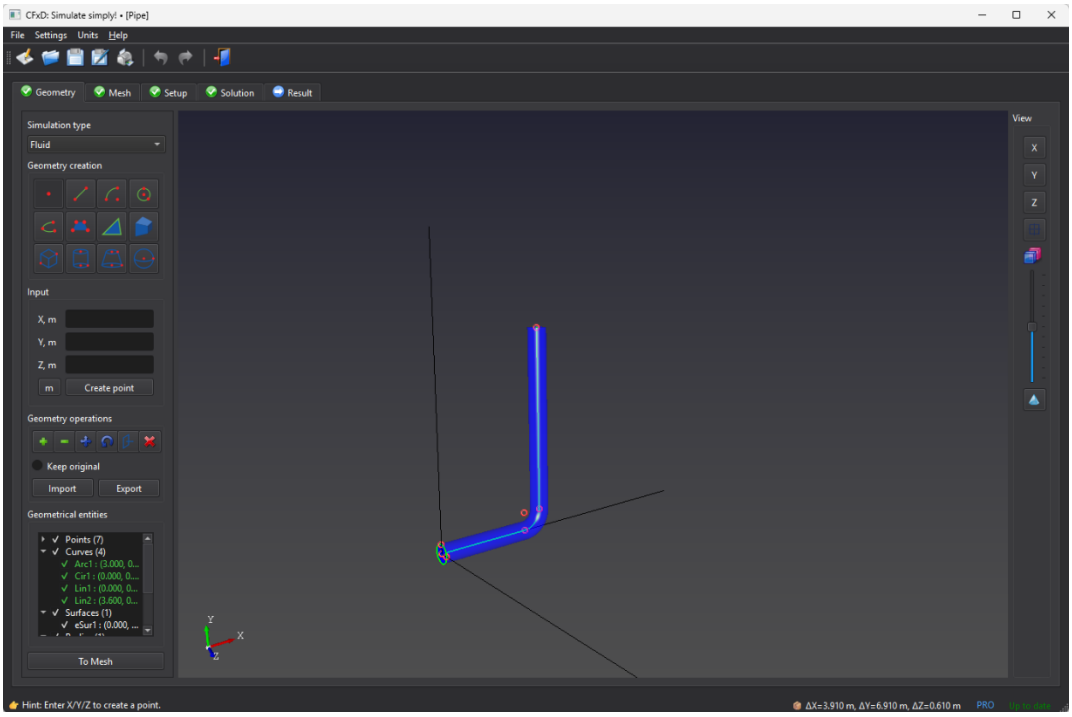


Figure 11. Geometry Tab.

The origin of the coordinate system (0, 0, 0) is located at the center of the graphical window. The axes are visually represented by the black axis in the graphical window with designated X, Y, Z coordinates shown at the bottom-left corner.

The status bar in the bottom provides helpful hints for geometry creation at the left, while overall geometry dimensions are displayed on the right.

3.1 Mouse manipulation

The software supports a standard three-button mouse with a scroll-wheel (Figure 12) for geometry creation and manipulation.



Figure 12. Three-button mouse.

The following notation will be used: **LMB** - the left mouse button, **MMB** - the middle mouse button, **RMB** - the right mouse button, **SWM** a scrolling wheel of a mouse. The mouse commands used in CFxD are introduced in Table 14.

Table 14: Commands used for the mouse buttons in CFxD.

Buttons	Action
LMB	select elements
MMB	rotate
SWM	in/out zoom
RMB	select for naming and pan

3.2 Geometry Creation

The basic geometries can be created using the following basic geometric primitives: point, line, arc, circle, curve, surface, cube, cylinder, cone and sphere (Figure 13).

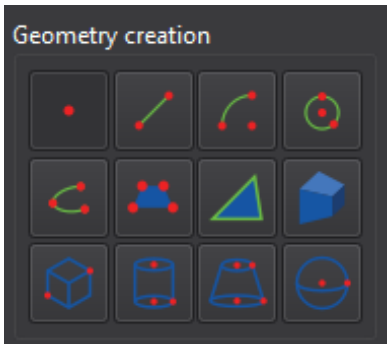


Figure 13. The geometry creation buttons.

A summary of all geometric primitives, their creation methods, and limitations is provided in Table 15.

Table 15: Geometrical primitives with properties.

Geometry	Icon	Creation	Limitation
Point		spatial coordinates: x, y, z	---
Line		by two (2) points selection	distinct points
Arc		by three (3) points selection 1: centre, 2: on radius, 3: on radius	two points on same radius, not collinear
Circle		by three (3) points selection 1: centre, 2: on radius, 3: on radius	two points on same radius, not collinear
Curve		by three (3) points selection	distinct points
Surface		by four (4) points selection	coplanar, ordered not collinear
		by three (3) curve selection	closed loop
Body		by six (6) surfaces selection	closed volume
Box		by two (2) points selection	opposite corners, axis-aligned
Cylinder		by three (3) points selection	radius ⊥ height
Cone		by four (4) points selection	both radii ⊥ height
Sphere		by two (2) points selection	distinct points

The following colours are used for geometry entities: red circles for points, green for edges, blue for surfaces and gray for solids, light blue for geometry highlighted by hovering mouse pointer, white for selected geometry.

Point creation

The **X**, **Y**, and **Z** fields in the **Input panel** define a point • in 3D space (Figure 14). Units can be switched using the **Units** button m (e.g., **mm** for SI or **in** for Imperial). Click **"Create Point"** button to add the point to the graphic window. Points are displayed as small **red** markers; a **selected** point is highlighted in **light blue**. Coordinate-based point entry is available also while creating other geometry primitives.

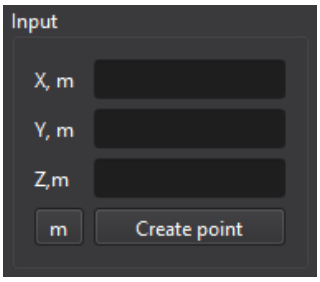


Figure 14. Input panel.

The point can be selected using LMB and deselected by a mouse click anywhere on the screen outside of the point. An example of some created points is shown in Figure 15.

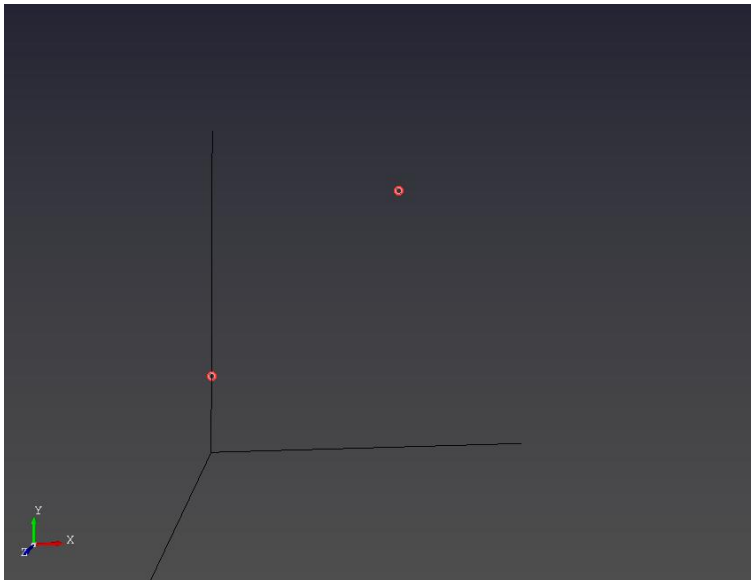


Figure 15. Created points.

Line creation

A **line** can be created by selecting two points using the **LMB**. An example showing a created line is presented in Figure 16. Like points, a line can be selected using **LMB** and deselected by mouse click anywhere on the screen outside the selected element.

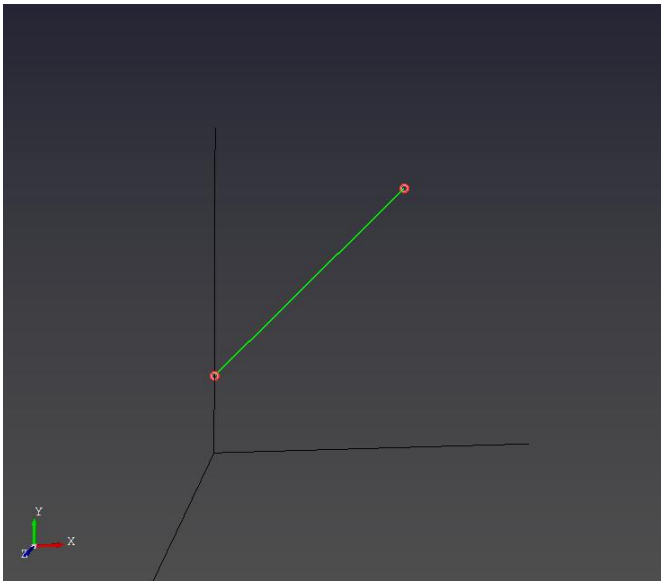


Figure 16. Line created with two points selection.

Arc creation

An **arc** can be created using three points**,** which must be selected in the following order**:** centre point (1), first point on the radius (2), and second point on the radius (3). The selected points must not be collinear, and two radius points must be on the same circle. An example of a created arc is shown in Figure 17.

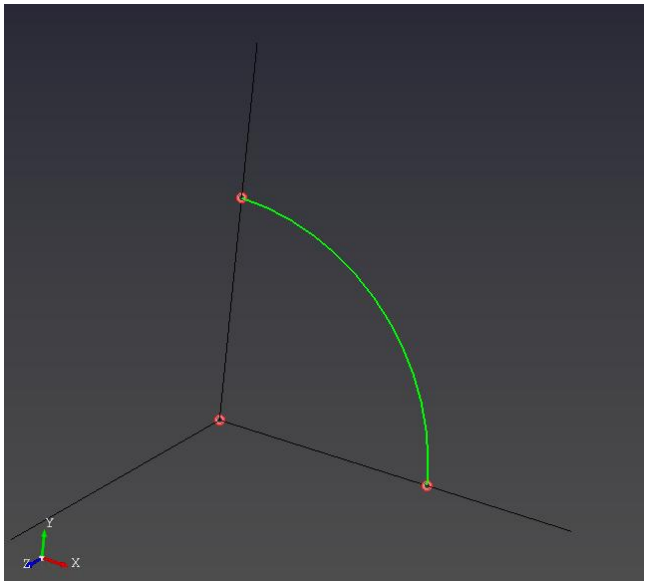


Figure 17. Arc created with three points selection.

Circle creation

Similar to an arc, a **circle** can be created using three points. The points must be selected in the following order: centre point (1), first point on the radius (2), and second point on the radius (3). The selected points must not be collinear, and the two radius points must be on the same circle. An example of a circle is shown in Figure 18.

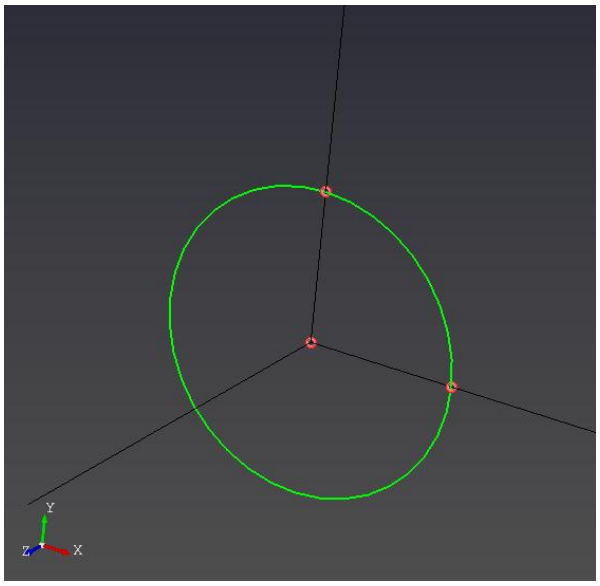


Figure 18. Circle created with three points selection.

Curve creation

A **curve** creation is based on three points (Figure 19). B-spline is used to get a curve going through selected three points.

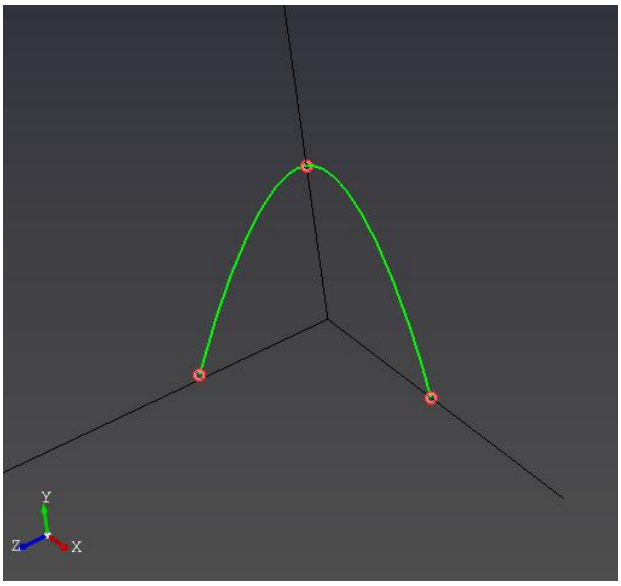


Figure 19. Curve created with three points selection.

Surface creation

Surface creation supports two methods: using four (4) points, or selecting three (3) lines or curves.

For the first method, exactly four distinct points must be selected, and they must define a closed boundary. Select the points sequentially—clockwise or counterclockwise—so that four connecting edges are implicitly formed. A surface cannot be created by selecting two diagonal pairs of points, because this does not define a valid boundary; the four points must follow one another along the perimeter of the surface to ensure proper surface generation. When creating a surface from points, all points must lie on the same plane. If the four points are not coplanar, surface creation will fail.

When creating a surface from lines or curves, the boundary must be defined by exactly three (3) lines or curves. A circle is a special case: a surface can be created from a single circular curve because it already forms a closed loop. In all cases, the selected lines and curves must form a closed loop and lie on the same plane. The examples of created surfaces are shown in Figure 20 (left: created by points, right: created by lines).

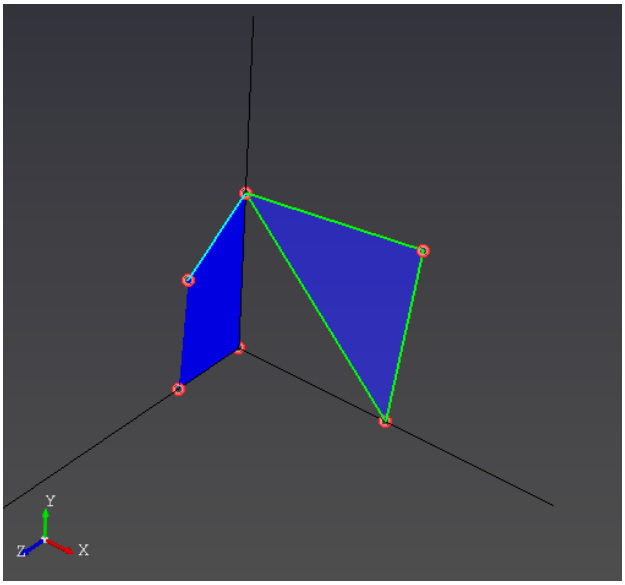


Figure 20. Created surfaces by points (left) and by lines (right).

Face division with a line or closed loop

CFxD can divide an existing planar face using either a line or a closed loop. Face division creates separate boundary faces while preserving the original solid volume. This is useful when different boundary conditions, material assignments, interface definitions, or local mesh settings are required on different portions of the same geometric face.

A line divider creates a partition across the selected face. The line must lie on the face and define a valid division within its boundary. A closed-loop divider creates an internal face region enclosed by the loop. The loop may be formed from connected geometric elements such as lines, arcs, circles, or curves. The selected divider must be planar with the target face, must not self-intersect, and must remain within the valid face region. Division fails if the line or loop is outside the face, crosses an existing hole or partition incorrectly, is non-planar, or does not define a valid boundary. Face division does not create a new solid and does not change the volume of the body. It only subdivides the boundary representation of the existing solid. The resulting faces can be selected independently, assigned different names, and used for boundary-condition or mesh definitions. The example is shown in Figure 21.

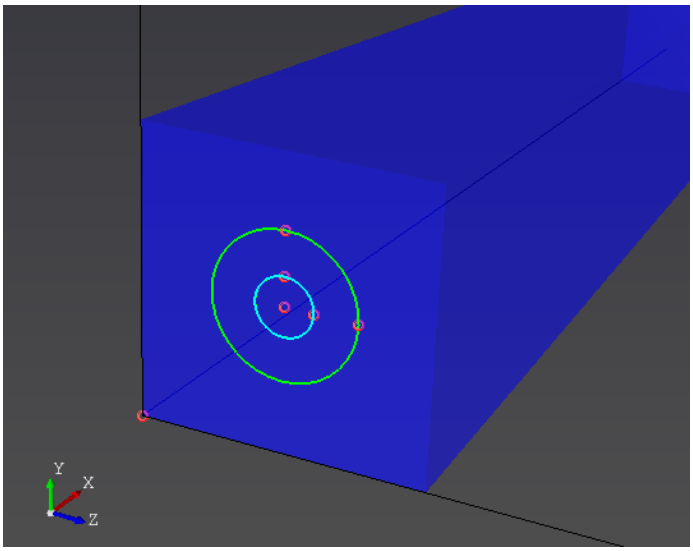


Figure 21. Face division with two closed loops.

A divider can be removed later through the geometry tree or the corresponding delete operation. Removing the divider merges the affected partitions back into the original face when no other geometric feature prevents the merge.

Box

A **Box** is created from two opposite corners. Faces are aligned with the global axes (parallel to XY, XZ, YZ planes). If the two points have the same X, Y, or Z coordinate (zero size in one direction), an error is shown. An example of a created box is shown in Figure 22.

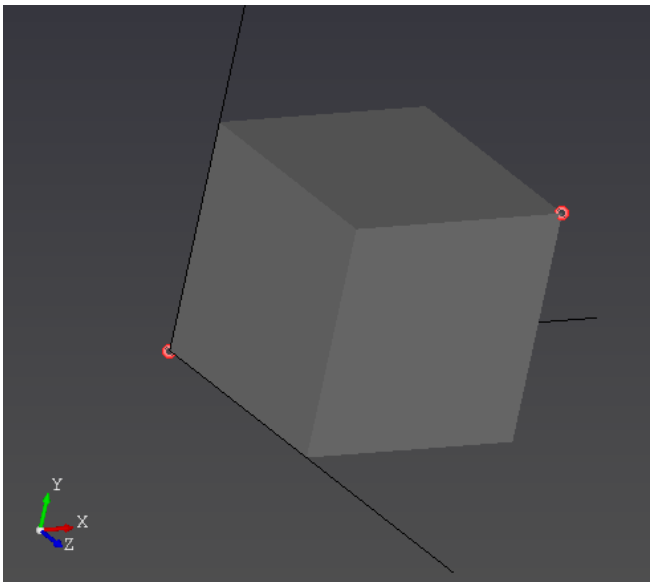


Figure 22. Box created with two points selection.

Cylinder

A **cylinder** can be created by selecting three points: (1) base center, (2) a point on the radius, and (3) a point defining the height direction and length. The radius direction (1→2) must be perpendicular to the height direction (1→3). An example of cylinder is shown in Figure 23.

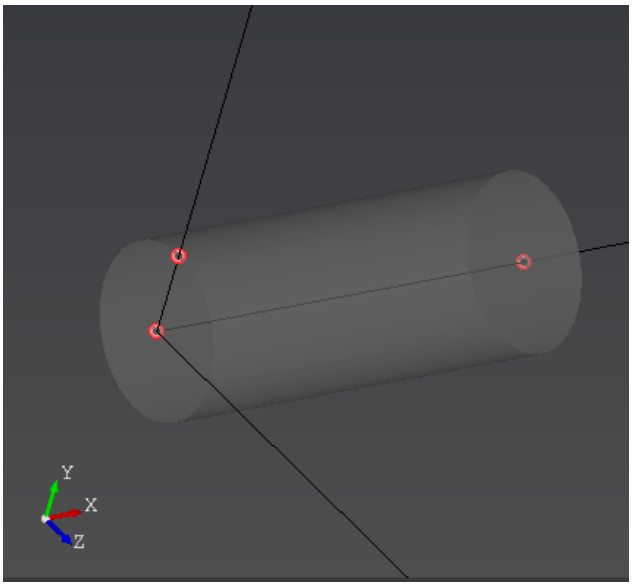


Figure 23. Cylinder created with three points selection.

Cone

A cone (frustum) can be created by selecting four points: (1) bottom center, (2) bottom radius point, (3) top center, (4) top radius point. The height direction is (1→3). Both radius directions (1→2 and 3→4) must be perpendicular to the height direction. An example of cone is shown in Figure 24.

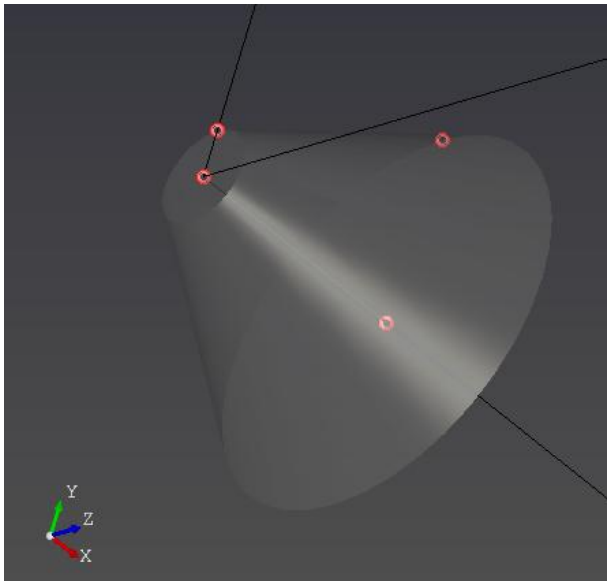


Figure 24. Cone created with four points selection.

Sphere

A sphere can be created by two points selections: central point and point on radius. An example of sphere is shown in Figure 25. Sphere is generated with upper and bottom faces.

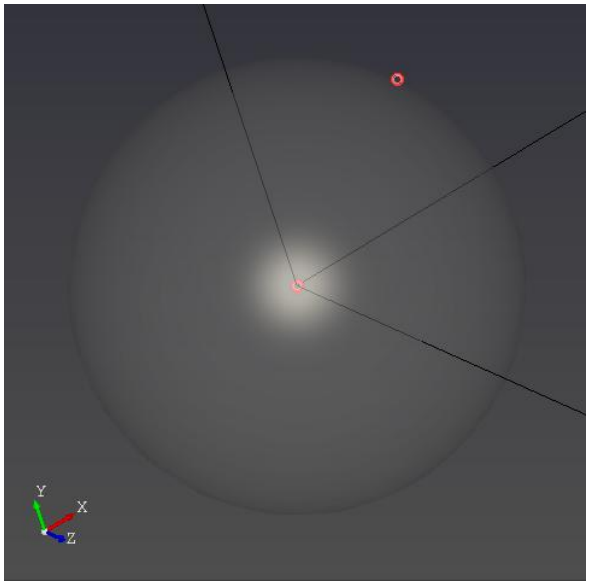


Figure 25. Sphere created with two points selection.

Body

Body can be created by selecting six (6) faces that form the closed body. The example of body is shown in Figure 26. It has been created by following operations: 8 points → 6 surfaces → body.

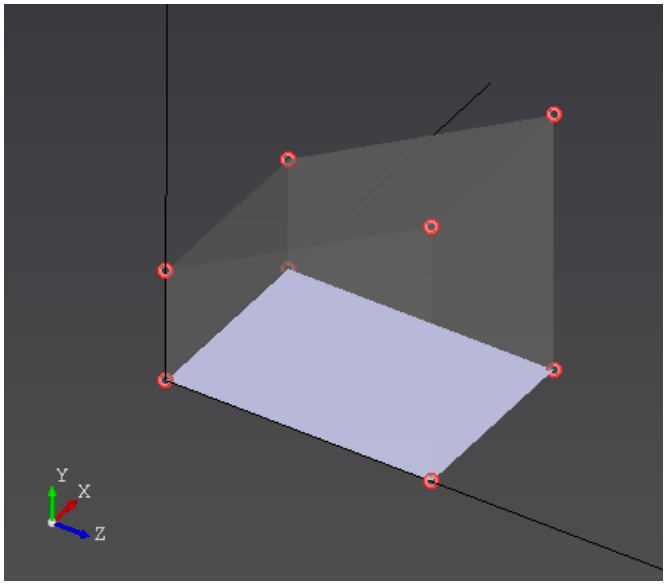


Figure 26. Body created with six surface selection.

3.3 Geometry operations

The **Geometry Operations** panel provides tools for

- Boolean operations (**Combine** and **Subtract**),
- Geometry relocation (**Translate** and **Rotate**),
- **Pull**,
- **Delete**.

The Boolean operations require the selection of two bodies. The **Input panel** (Figure 27) is used to specify translation distances, pull distance and rotation angles. The **Pull** operation works only with planar surfaces. In addition to pulling in the normal distance to surface, it also provides an operation to pull along a selected curve.

For Boolean and relocation operations, the original geometry can be preserved by enabling the **Keep original** option. If this option is not enabled the original geometry is removed after the operation completes.

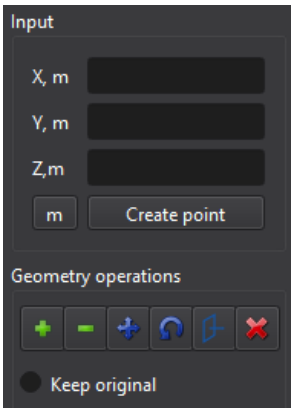


Figure 27. Geometry operations panel.

3.4 Geometry import

Geometry created in other programs can be imported into CFxD using the **Import** button located below the **Geometry Operation** panel. Currently, only the **STEP** format is supported. CFxD automatically converts the geometry dimensions from the STEP file. An example of imported geometry is shown in Figure 28.

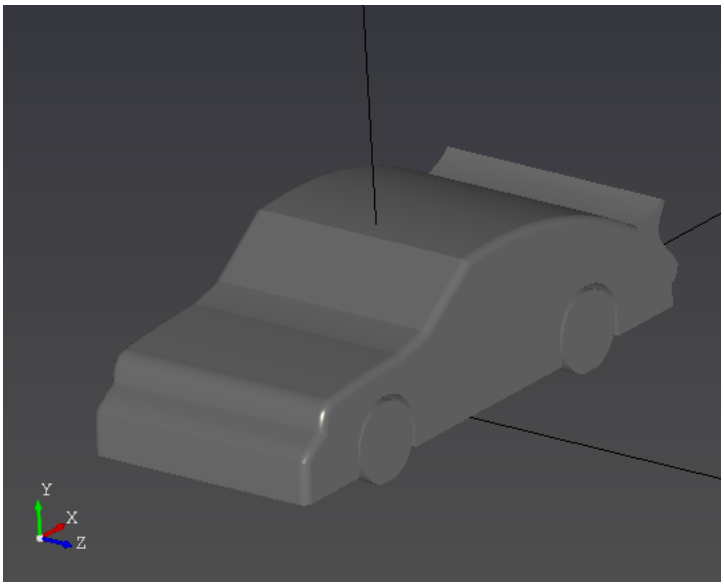


Figure 28. Imported geometry.

3.5 Geometrical entities window

The created geometric entities appear in the Geometrical entities window (Figure 29) located below Geometry operation panel. This window is updated automatically when entities are created or deleted. The entities are categorized in the following groups: Points, Curves, Surfaces, Bodies and Named faces. Pressing the box with the checkmark will hide/unhide geometry or category. Geometry can also appear highlighted in the graphical window by clicking the desired geometry in the geometrical entities window.

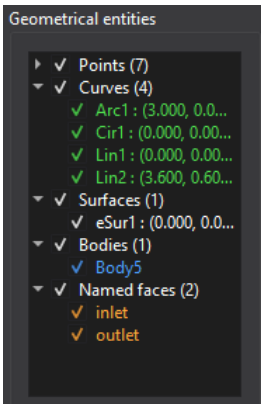


Figure 29. Geometrical entities window.

Deleting/Renaming geometry or name

Geometry or name may be deleted or renamed from the geometrical entities window by clicking on desired entity with RMB and selecting **Delete** (Figure 30).

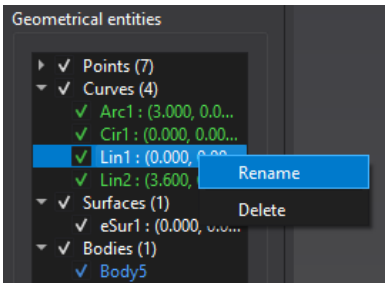


Figure 30. Deleting Body1 in geometrical entities window.

3.6 View settings

Basic **view settings**, such as **views from X, Y, Z** directions, **fit all**, **wireframe/solid**, **and transparency control**, are located to the right of the graphic window (Figure 31).



Figure 31. View settings.

3.7 Naming

The naming operation can be activated in the Graphic window with RMB. RMB is used to select a face to assign a name in the appeared dialog (Figure 32). By default, the names beginning with “inl” will be recognized as inlets during the Setup step. Similarly, “out” is recognized as an outlet, “wall” as a wall, “sym” as a symmetry boundary, and “inter” as an interface. The interface should be an internal surface.

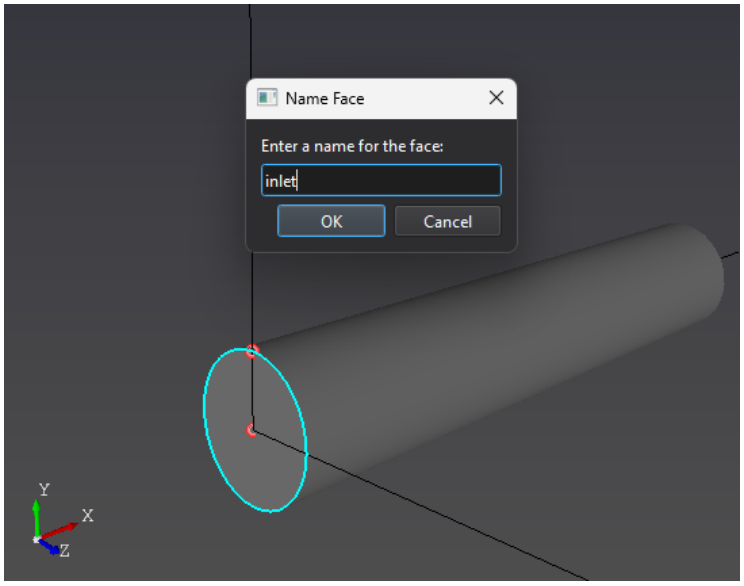
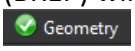


Figure 32. Face naming.

3.8 Export geometry

The last action in the **Geometry Tab** is export geometry to Mesh. The **"To Mesh"** button located at the bottom left menu will convert the geometry to the format (STL) comparable to the mesh generator. It will be saved in the Mesh folder. Only a solid element that forms a closed volume can be converted to the format for future meshing. The geometry in native format (BREP) will be saved in the folder Geometry as well. After successful export the **Geometry Tab** 

will be changed to **Mesh Tab** 

automatically.

Chapter IV. Mesh

The mesh settings, information, and graphical representation of the mesh are shown in Figure 33. Currently, CFxD supports **hexahedral-dominant** mesh. The mesh generation process includes the following settings: base **cell size**, local controls for **surface/volume refinement** and **zones**. **Surface** and **Volume refinement** dialogs can be called with RMB on **Mesh viewer** screen or from left menu Surface Volume. Initially, a recommended cell size is **automatically calculated** based on the overall dimensions of the geometry.

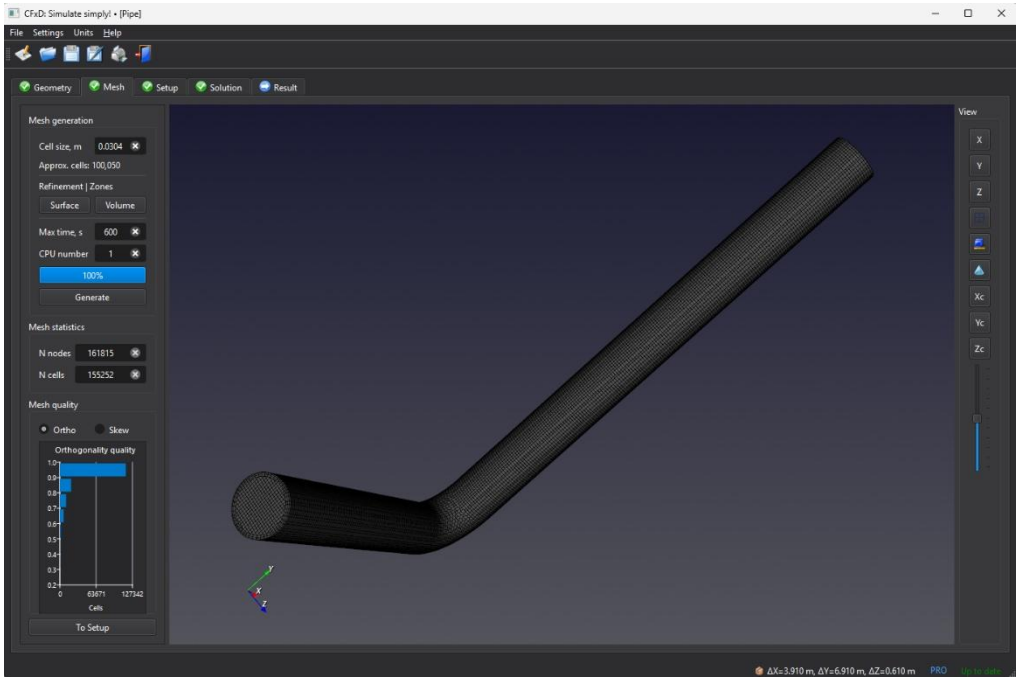


Figure 33. Mesh tab.

4.1 Refinement

The Surface mesh settings dialog allows local mesh controls to be applied to selected surfaces (Figure 34). The following options are available: **Refinement**, **Boundary layers**, **Thin wall**, and **Porous**.

Refinement allows the local cell size to be specified on a selected surface. **Boundary layers** add prism layers to better resolve near-wall gradients. The number of layers, first-layer thickness, and growth ratio can be specified, while the total layer thickness is calculated automatically. The **Thin wall** option allows an internal surface to be represented as a wall without explicitly modeling its thickness. The **Porous** option identifies an internal surface for porous-jump modeling, where the pressure loss is defined during the Setup stage.

The Preview button highlights the selected surface in the Mesh viewer. When refinement is enabled, a blue cell-size cube indicates the specified local cell size. The Add button adds the selected surface and its settings to the table. Multiple surfaces with different settings can be defined. Existing entries can be deleted by right-clicking (RMB) the corresponding row in the table.

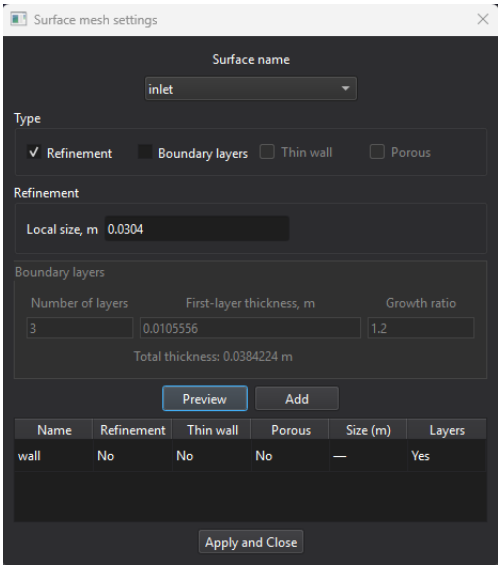


Figure 34. Surface mesh refinement dialog.

In addition to surface refinement, **volume refinement** can be used. Two predefined volume shapes are available: **box** and **cylinder**. The box is defined by two diagonal points, and a local cell size can be specified for the cells inside the box. A red box with a blue cell-size cube is shown when **Preview** is clicked (Figure 35).

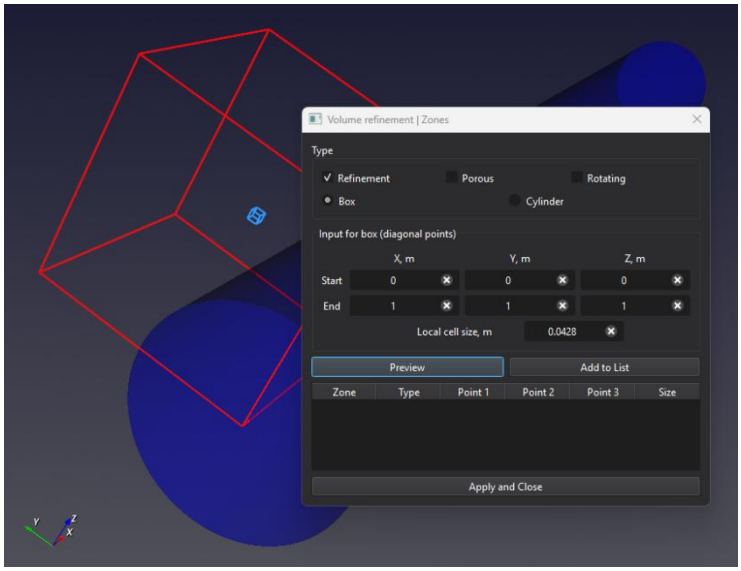


Figure 35. Volume mesh refinement dialog with a box.

Similarly, refinement inside a cylinder can be added. The cylinder is defined by three points: the center, a point defining the radius, and a point defining the height. Added volumes (box/cylinder) can be deleted with the right mouse button (RMB) in the table in the **Volume refinement** dialog. The volume refinement dialog can be used to mark the zone that can be used for porous structure (Figure 37) or rotating domain.

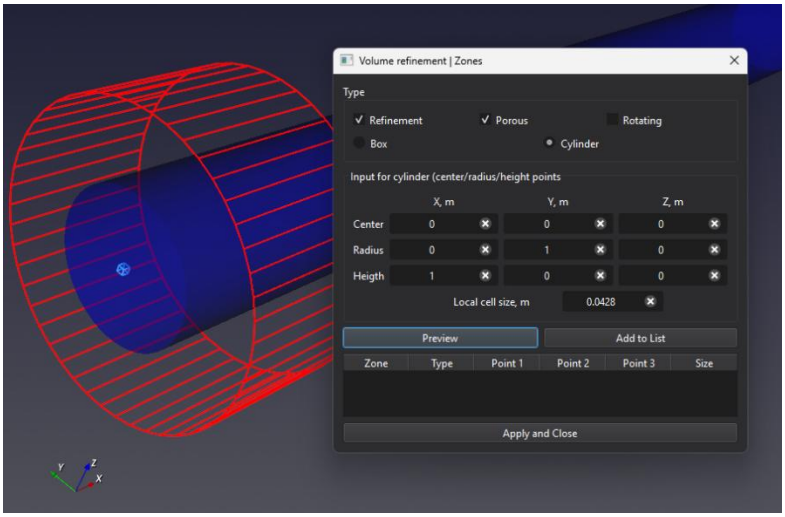


Figure 36. Set a cylinder for refinement and as a porous zone.

The example of mesh refinement with box and cylinder is shown in Figure 37.

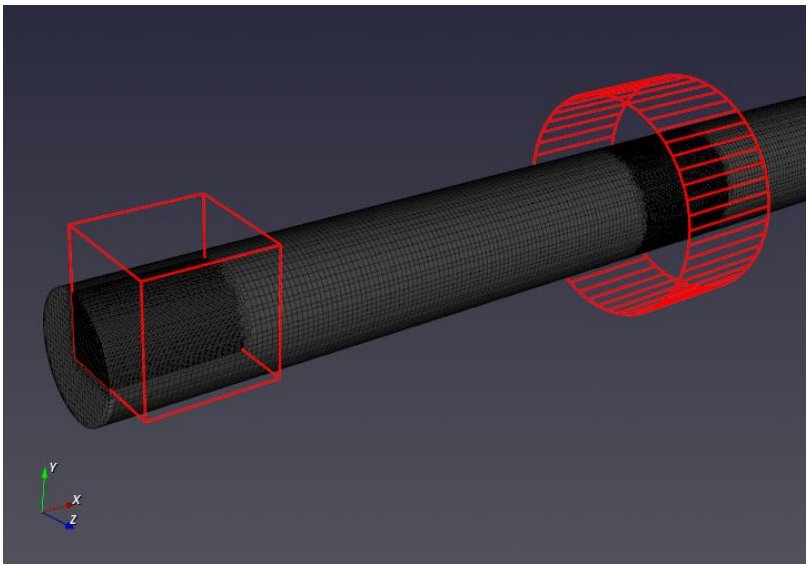


Figure 37. Example of volume refinement.

The mesh generation process can be limited by a **maximum execution time**. Mesh generation starts using the **Generate** button, and the progress of the process is displayed in a **progress bar**.

4.2 Mesh statistics, Quality and Export

The **Mesh statistics** panel is located below the **Mesh generation** panel. It provides summary information for the generated mesh, including the total number of nodes and cells (Figure 38).

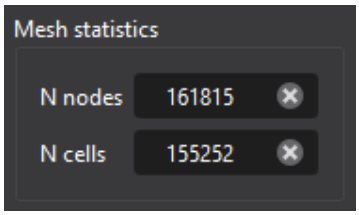


Figure 38. Mesh statistics panel.

The **Mesh Quality** panel shows a histogram of mesh quality metrics such as **orthogonality** and **skewness** (Figure 39). The metric can be selected using the **Ortho** or **Skew** option. The histogram shows the distribution of cells across different quality ranges, allowing the overall mesh quality to be evaluated.
Cells within a selected quality range can be highlighted in the Mesh viewer by clicking the corresponding histogram bar. This allows regions containing poor-quality cells to be identified and inspected visually.

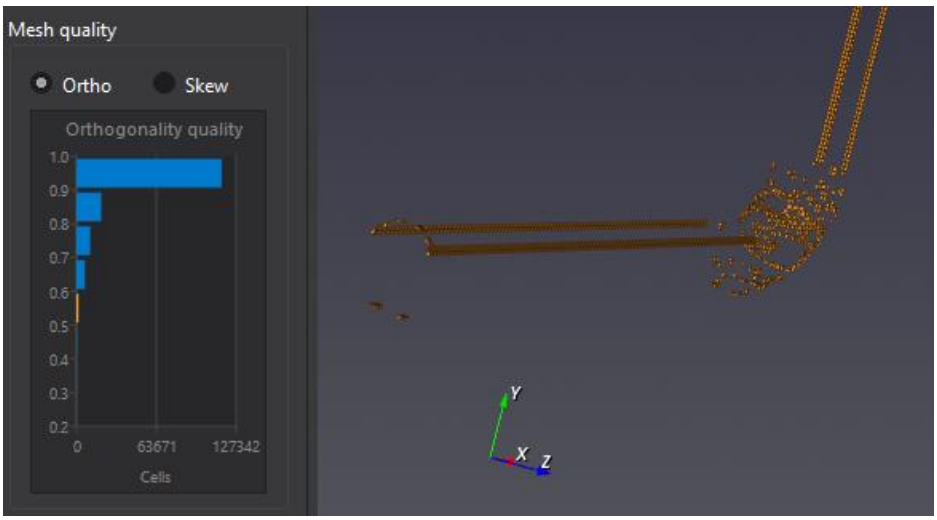


Figure 39. Mesh quality panel.

Similar to the **Geometry** tab, the **Mesh** tab includes a **View** panel with the same navigation and display options. In addition, it provides mesh-specific features, including a coordinate scale, that can be shown or hidden using the **Scale** button in the **View** panel (Figure 40).

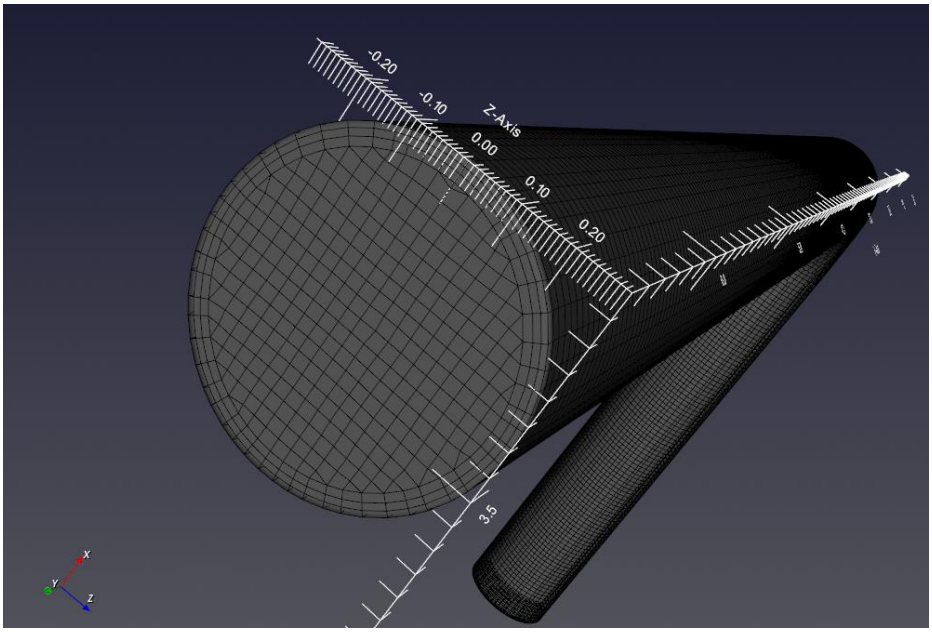


Figure 40. Coordinate scale.

For visual inspection of the internal mesh, sectional cuts can be displayed in the **Xc**, **Yc**, and **Zc** planes using the cut controls and slider (Figure 41).

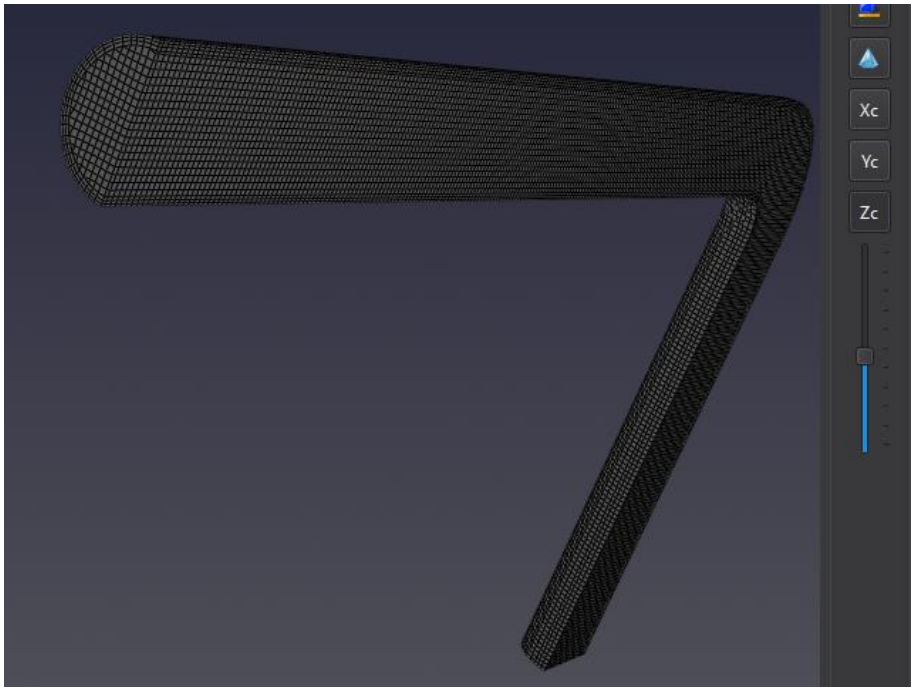


Figure 41. Cut section.

By pushing button “To Setup” the mesh will be exported to Setup step and **Mesh** Tab will be changed to **Setup** Tab.

Chapter V. Setup

The solver settings and boundary conditions are defined in the **Setup** tab (Figure 35). This tab is used to configure the physical model and numerical setup before running the simulation. It allows:

- selection of the **simulation mode** (steady or transient) and the corresponding iteration/time settings,
- selection of the **flow regime** (laminar or turbulent) and the **turbulence model** (when applicable),
- selection of the **flow type** (**incompressible** or **compressible** subsonic flow);
- enabling or disabling **heat transfer**,
- definition of **operating conditions** (e.g., reference pressure and gravity/buoyancy),
- selection of **materials** and assignment of thermophysical properties to domains,
- assignment of **boundary conditions** to named faces (inlet, outlet, wall, symmetry, interface),
- selection of the number of **CPU cores** for parallel execution.

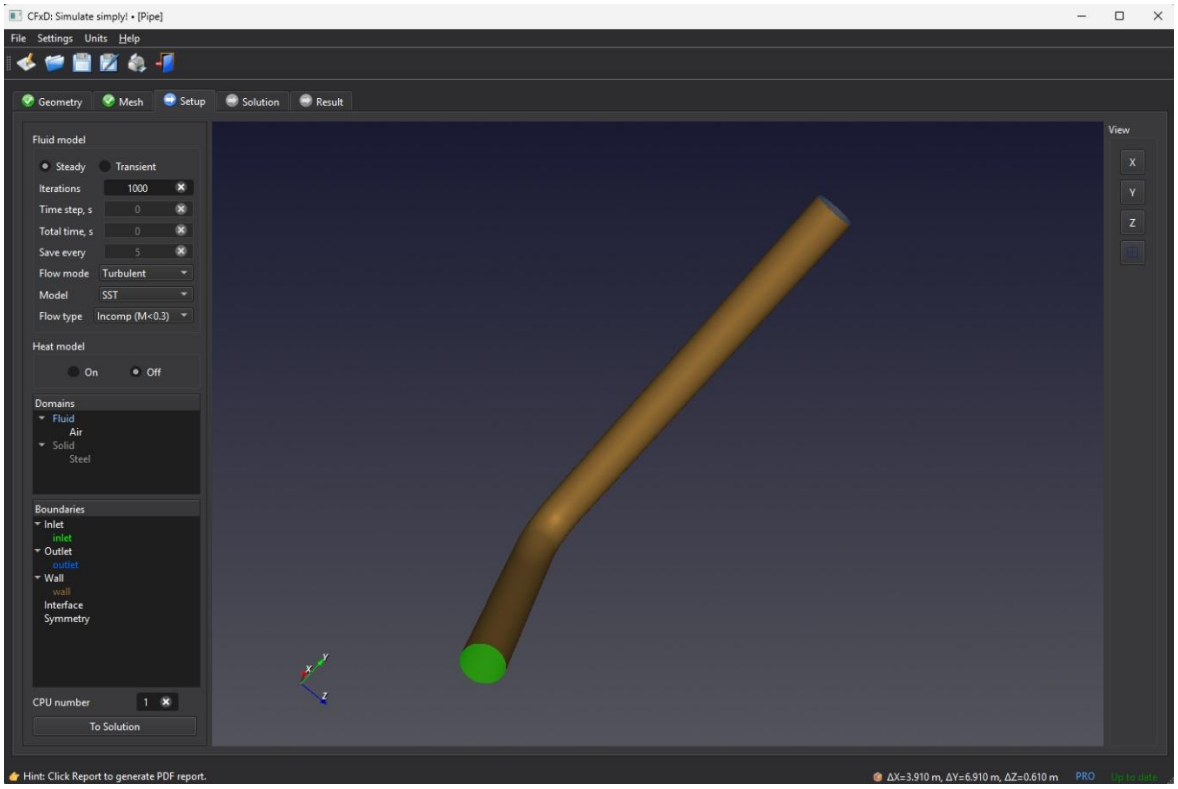


Figure 42. Setup tab.

5.1 Physics model

The **Physics model** panel displays a summary of the currently selected physical models. A short description of the active configuration is shown below the model list. Button **Edit…** opens the Physics dialog, where the flow, heat-transfer, multiphase, species, combustion, and acoustics models can be configured. The selected settings are shown in the panel after the changes are applied (Figure 43).

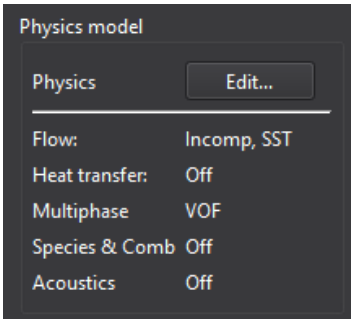


Figure 43. Physics model panel.

The Physics dialog is organized into the categories **Flow**, **Heat transfer**, **Multiphase**, **Species & Combustion**, and **Acoustics** (Figure 44). Each category from the list allows to view and configure its available models. The selected settings are applied using **Apply and Close**.

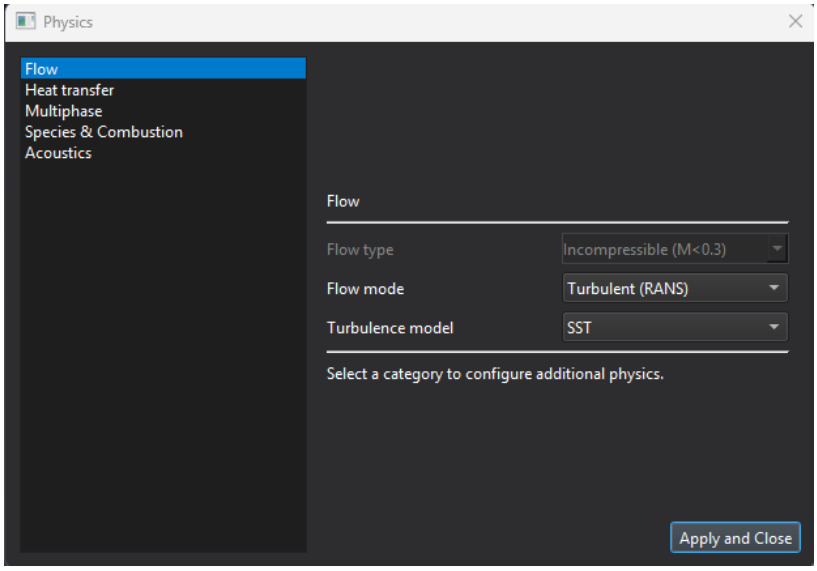


Figure 44. Physics dialog.

Flow

The **Flow** category defines the basic flow formulation. **Flow type** specifies the compressibility model:

- **Incompressible** for low-speed flow, typically (M<0.3);
- **Compressible Sub** for subsonic compressible flow, (M<1);

- **Compressible Hi** for high-speed flow, (M≥1).

Flow mode specifies whether the flow is modeled as:

- **Turbulent (RANS)**;
- **Laminar**.

For turbulent flow, the available turbulence models are **SST** and **k–ε**.

Heat transfer

The **Heat transfer** category enables or disables the energy equation. When heat transfer is enabled, temperature is solved together with the flow variables. This option is required for compressible flow, gas mixtures, combustion, and conjugate heat-transfer simulations.
The radiation model can be set to:

- **Off**;
- **S2S**, for surface-to-surface radiation between wall surfaces.

Multiphase

The **Multiphase** category enables the **Volume-of-Fluid (VOF)** model. VOF is intended for two immiscible fluids sharing one fluid domain. Enabling VOF selects incompressible isothermal flow and disables incompatible models such as heat transfer, gas mixtures, combustion, acoustics, porous models, rotating zones, and fan interfaces.

Species and Combustion

The **Species & Combustion** category provides models for gas mixtures and reacting flow. The **Mixture** option enables gas-mixture species transport. The composition of the mixture is defined using the species and mass fractions specified for the materials and boundary conditions. The **Combustion** option enables reacting-flow calculations. CFxD currently provides the **Methane–Air** reaction mechanism. For laminar flow, finite-rate Arrhenius chemistry is used. For turbulent flow, the Eddy Dissipation Model is used. Gas-mixture and combustion models require a fluid simulation and may require compressible flow and heat transfer. They are not compatible with VOF.

Acoustics

The **Acoustics** category provides the **Proudman** broadband acoustic model. It estimates acoustic power generated by isotropic turbulence from the flow and turbulence fields. Proudman acoustics requires turbulent flow and is not available with VOF. For incompressible flow, the reference sound speed is specified in the Physics panel. For compressible and thermally coupled cases, the density and sound speed are obtained from the fluid thermodynamic state. The model produces acoustic power density and acoustic power-density level at solver output times. It does not calculate acoustic-wave propagation or microphone sound-pressure level.

The availability of individual models depends on the selected license, simulation type, and compatibility with the other physics settings. Incompatible options are disabled or reset automatically when the Physics configuration is applied.

5.2 Run settings

The **Run settings** panel defines whether the simulation is steady or transient and controls the iteration, time-step, and output settings (Figure 45).

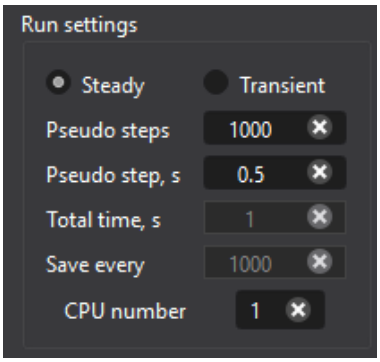


Figure 45. Run settings.

Steady simulation

The available fields for steady simulation are:

- **Iterations** - the maximum number of solver iterations;
- **Save every** - the interval, in iterations, at which results are written.

For steady gas-mixture and high-speed cases, the iteration field is shown as **Pseudo steps** and the time-step field is shown as **Pseudo step**. These quantities represent pseudo-time advancement used to obtain a steady solution and do not represent physical time.

For steady VOF cases, the time-step field is shown as **Max local step**. The solver uses local time stepping, and the specified value limits the local pseudo-time step. The **Save every** field specifies the number of local-time-stepping iterations between output times.

Transient simulation

Select **Transient** when the time-dependent behavior of the flow must be resolved. The available fields are:

- **Iter. per step** — the maximum number of solver iterations performed within each physical time step;
- **Time step** — the physical time-step size;
- **Total time** — the total physical duration of the simulation;
- **Save every** — the number of time steps between saved results.

Transient operation is required for time-dependent phenomena such as oscillating flows, waves, sloshing, filling, draining, transient rotor–stator interaction and evolving multiphase interfaces.

The available fields change automatically according to the selected physics model. For example, VOF cases use interface-stability controls in addition to the flow Courant-number limit, while steady gas-mixture and high-speed cases use pseudo-time settings.

The flow regime, turbulence model, compressibility, heat transfer, multiphase model, combustion, and acoustics are configured separately in the **Physics** panel described in Chapter 5.1.

Parallel settings

At the bottom of the **Run Settings**, the number of **CPU cores** can be selected to run the simulation in parallel. Parallel execution can significantly reduce runtime for medium and large cases, depending on mesh size and solver settings.

If the requested number of cores exceeds the number of available CPU cores, CFxD automatically reduces it to the maximum supported value. The selected core count is applied during the **Solution** step when the case is decomposed, and the solver is launched in parallel. After the run, CFxD reconstructs the results for post-processing.

For small meshes, using many cores may provide limited benefit due to parallel overhead. CFxD displays a notification if an excessively large number of CPU cores is selected for a small mesh.

5.3 Operating conditions and buoyancy

Operating conditions are configured via **Top Menu → Settings → Operating conditions** (Figure 46). This panel is used to define the **reference pressure** and **gravity vector**, which are required for consistent pressure specification and buoyancy-driven flow modeling.

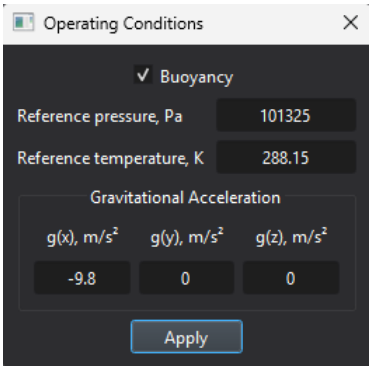


Figure 46. Operating conditions panel.

Reference pressure. The reference pressure is used as the ambient (baseline) value for cases where pressure is specified as **gauge pressure** in the boundary-condition panels. For compressible subsonic simulations, CFxD uses the reference pressure to convert gauge pressure values to absolute pressure internally.

Reference temperature. The panel also includes a **reference temperature** field. In the current version, this value is displayed for completeness and future extensions, but it is **not used** in the solver setup and does not affect the simulation results.

Buoyancy (natural convection). Natural convection can be enabled by activating the **Buoyancy** option in the same panel. When buoyancy is enabled, gravity introduces a density-driven body force in the momentum equation based on temperature-dependent density effects. Buoyancy is typically used for problems such as heated enclosures, vertical plumes, and thermally driven circulation.

Gravitational acceleration. The gravity vector components **g(x)**, **g(y)**, and **g(z)** define the direction and magnitude of gravity. For example, a vertical gravity field is commonly set by assigning -9.81 m/s^2 to the appropriate component aligned with the vertical axis of the model.

5.4 Domains

Materials are assigned to computational domains through the **Domains** tree. Select a domain and open the **Domain properties** dialog to assign the corresponding fluid or solid material (Figure 47). Common materials are available in the material library, and additional materials can be created or edited through the Materials panel.

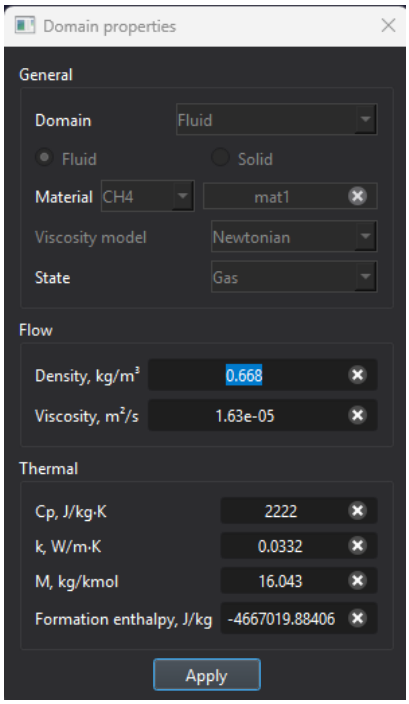


Figure 47. Material selection.

For a single-fluid simulation, one fluid material to the fluid domain must be assigned. The selected material provides the density, viscosity, thermal properties, and other properties required by the selected physics models.

For VOF simulations, two fluid materials to the fluid domain as **Phase 1** and **Phase 2** should be assigned. The two phases are used by the volume-fraction model to calculate the local mixture properties. The surface-tension coefficient is specified in the VOF physics settings.

For gas-mixture simulations, the component gases used by the mixture must be selected. The species properties are obtained from the selected materials and are transported according to the mixture model.

For combustion simulations, the selected methane–air mechanism determines the required species. The corresponding species materials can be reviewed and edited under the fluid-domain properties. Their thermodynamic properties, transport properties, and formation enthalpies are used by the reacting-flow model.

For conjugate heat-transfer cases, assign fluid material properties to the fluid domains and solid material properties to the solid domains. For a solid domain, a volumetric heat source may also be specified. This represents uniform internal heat generation within the selected solid region and can be used to model heaters, electronic components, or other internally heated parts.

When porous zones are created, a **Porous** item is added to the Domains tree and the available porous zones are listed beneath it (Figure 48). Selection of a porous zone will open the **Porous zone** dialog. The porous zone is assigned to a volume region and configured using its thickness or reference length and directional resistance coefficients. CFxD supports both the Simple resistance model and the Viscous/Inertial model.

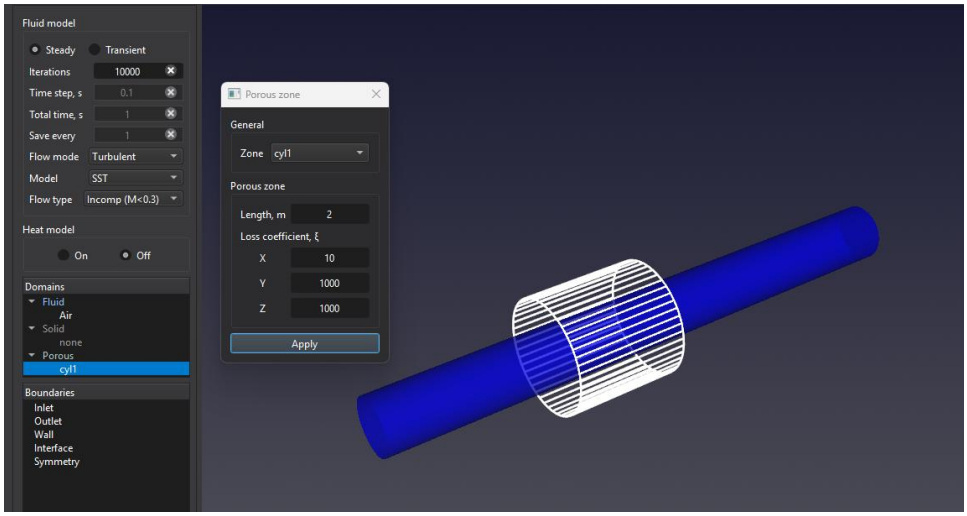


Figure 48. Porous zone setup.

When a rotating zone is created, a **Rotation** item is added to the Domains tree. The selected rotating zone is configured by specifying its rotation axis and rotational speed. The rotating-zone settings are used by the Multiple Reference Frame model.

For acoustics, the fluid-domain material supplies the density used by the Proudman model. In incompressible cases, the reference sound speed is specified separately in the Physics panel. In compressible and thermally coupled cases, the density and sound speed are obtained from the fluid thermodynamic state.

After changing the domain properties, the settings can be stored with **Apply**. The available domain properties depend on the selected simulation type and physics models. Incompatible options are disabled when the corresponding model is not active.

5.5 Boundary conditions

The **Boundaries** tree is used to assign boundary conditions to the named faces of the geometry. The available boundary types are **Inlet**, **Outlet**, **Wall**, **Symmetry**, and **Interface**. To distinguish the boundaries in the Viewer, inlet, outlet, wall, and symmetry faces are displayed using different colors. The following color-coding is used: inlet (green), outlet (blue), wall (gray) and symmetry (yellow).
The contents of the boundary-condition dialogs depend on the selected physics models. Controls that are not applicable to the current simulation are hidden or disabled.

Inlet

The **Inlet** dialog supports the following inlet types:

- **Velocity**;
- **Mass flow**;
- **Pressure**;
- **Open**;
- **Freestream**.

For a velocity inlet, the velocity components are specified directly. A mass-flow inlet uses the prescribed mass-flow rate. A pressure inlet specifies gauge pressure relative to the reference pressure. The Freestream option is intended primarily for external-flow calculations.
The inlet dialog may also include the following subpanels (Figure 49):

- **Thermal**, for specifying the inlet temperature;
- **Turbulence**, for specifying turbulent intensity and length scale;
- **Mixture**, for defining the inlet composition of a gas mixture;
- **VOF phase composition**, for specifying the phase-1 volume fraction;
- **Combustion**, for specifying species mass fractions.



Figure 49. Inlet input panel.

For gas-mixture cases, the inlet composition is defined using the mixture fractions of the selected gases. For VOF cases, the phase-1 volume fraction is specified between zero and one; the phase-2 fraction is obtained as the complementary value. For combustion cases, the species composition may be selected from a preset, such as air or fuel, or entered using custom species mass fractions.

Outlet

The **Outlet** dialog supports **Pressure**, **Open**, and **Mass flow** outlet types (Figure 50).

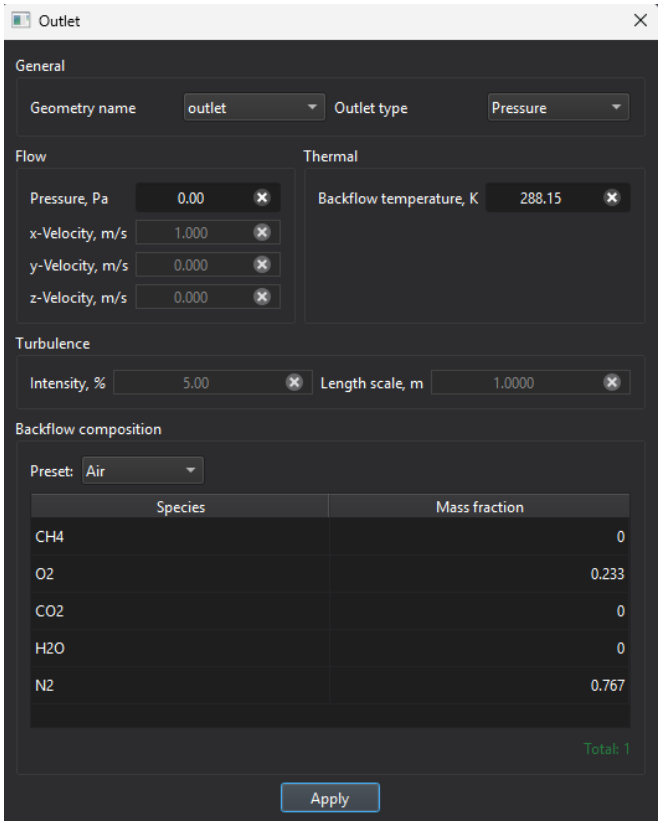


Figure 50. Outlet input panel.

A pressure outlet uses gauge pressure relative to the operating reference pressure. A mass-flow outlet specifies the required outlet mass-flow rate. An open outlet allows the flow direction to change and includes backflow values for velocity and temperature. Turbulence quantities may also be specified for reverse flow.
For VOF cases, the outlet dialog includes the phase-1 volume fraction used during backflow. For gas-mixture and combustion cases, the backflow composition can be defined using mixture fractions or species mass fractions.

Wall

The **Wall** dialog defines the mechanical and thermal behavior of a wall boundary (Figure 51).

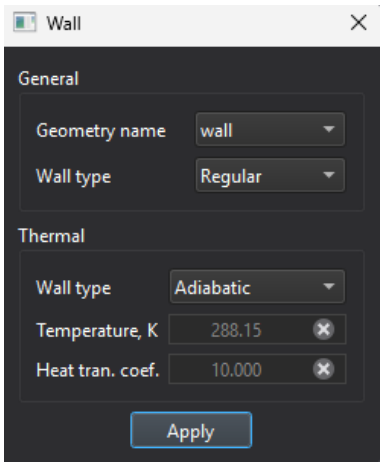


Figure 51. Wall input panel.

The available wall types are:

- **Regular**, representing a no-slip wall;
- **Frictionless**, representing a slip wall;
- **Rough**, representing a wall with specified roughness height.

The thermal condition can be selected as:

- **Adiabatic**;
- **Temperature**;
- **Convection**;
- **Heat flux**.

For convective walls, the ambient temperature and heat-transfer coefficient are specified. For a prescribed-temperature wall, the wall temperature is entered directly. Surface emissivity can be specified when the S2S radiation model is active.

For VOF cases, the **VOF/Wall adhesion** panel provides either zero-gradient phase-fraction behavior or a constant static contact angle. The contact angle is measured through phase 1.

Symmetry

The **Symmetry** boundary applies a symmetry condition to the selected face. Normal velocity and normal gradients of the appropriate transported quantities are constrained according to the symmetry condition.

Interface

The **Interface** dialog is used for internal faces that require a special coupling or pressure-loss model (Figure 52).

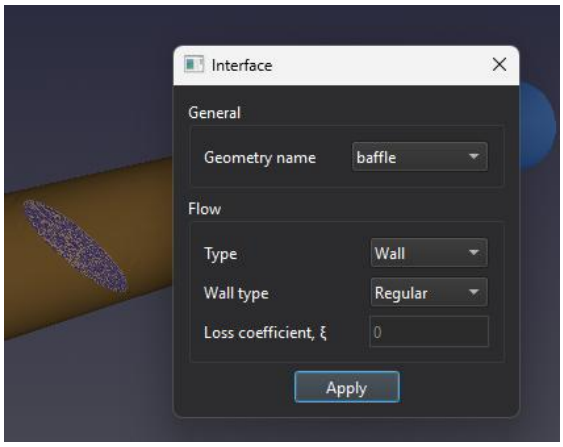


Figure 52. Interface input panel.

The available interface types are:

- **Thin wall**;
- **Porous**;
- **Fan**.

A thin wall can be configured as **Regular** or **Frictionless**. In CHT cases, the thermal interface can be assigned either **Perfect contact** or a specified **Thermal resistance**. For VOF cases, wall adhesion and a static contact angle may be assigned to the thin wall.

A porous interface represents a thin localized resistance. CFxD supports the **Simple** loss-coefficient model and the **Viscous/Inertial** model. The latter separates the linear viscous coefficient from the quadratic inertial coefficient.

A fan interface represents a simplified fan model. The pressure rise, blow direction, and optional swirl angle can be specified. When swirl is enabled, the fan center, hub diameter, and tip diameter define the annular region in which the tangential velocity contribution is applied. The fan interface does not resolve blade geometry or rotor motion.

After the boundary values are entered, **Apply** can be used to store the settings. Boundary assignments can be removed by selecting the corresponding item in the **Boundaries** tree, pressing the right mouse button, and choosing **Delete** (Figure 53).

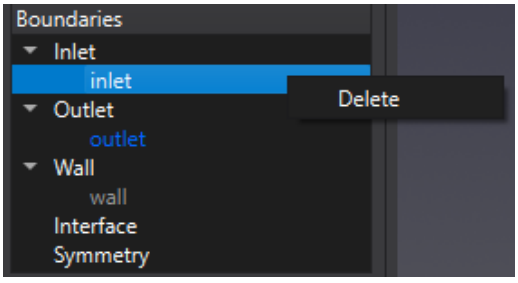


Figure 53. Deleting boundaries.

Chapter VI. Solution

Solution tab allows to set and monitor residuals (convergence plot), track field values at selected points or faces, adjust a stability control, and run/pause/resume a simulation (Figure 54).

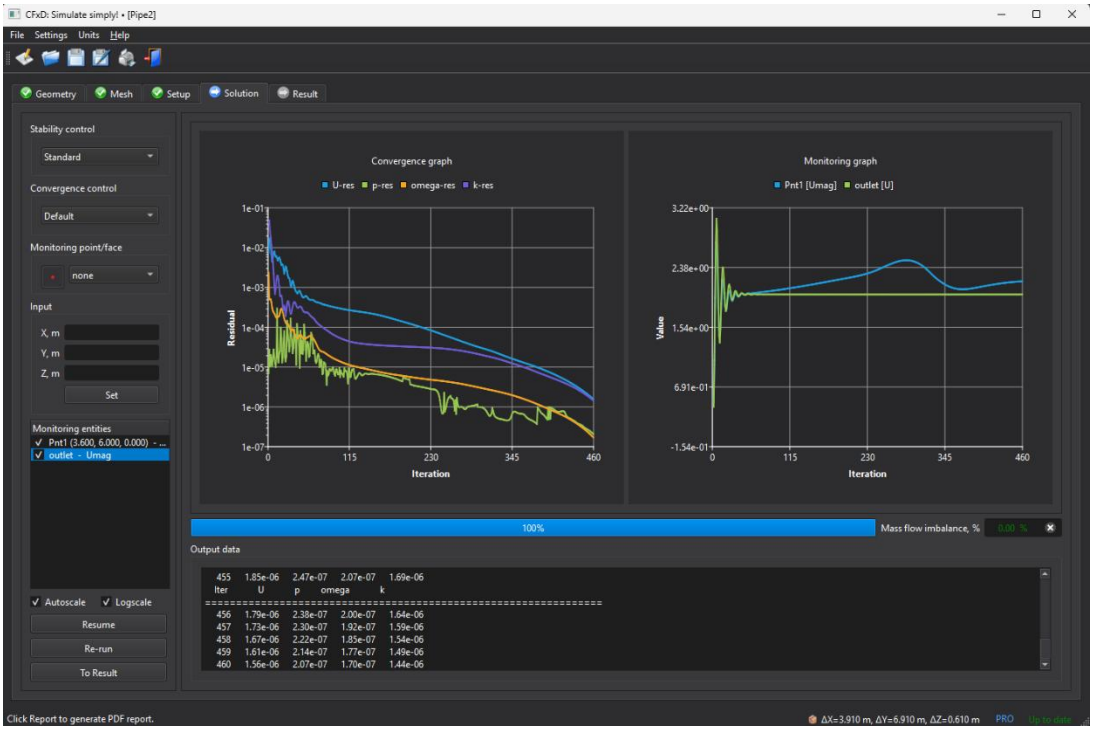


Figure 54. Solution tab.

6.1 Stability control panel

The **Stability control panel** (Figure 55) allows to adjust the relaxation factors to improve solution stability. It includes three preset modes: **Aggressive**, **Default**, and **Conservative**. **Aggressive** uses higher relaxation factors and may speed up convergence, but it can also make the solution unstable and lead to divergence. **Default** provides a balanced setting and is recommended for most cases. **Conservative** applies stronger damping (lower relaxation factors) to improve stability. It may converge more slowly but is useful for difficult meshes or highly nonlinear flows.

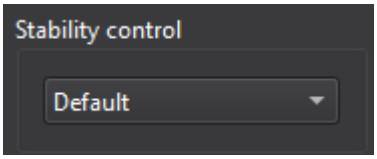


Figure 55. Stability control panel.

6.2 Monitoring point/face

The monitoring points or faces can be created in the **Monitoring point/face panel** (Figure 56). The points coordinates can be set in the **Input panel** similarly to points input in Geometry tab. The faces can be selected from the drop-down list. The list contains available boundary faces (patches).

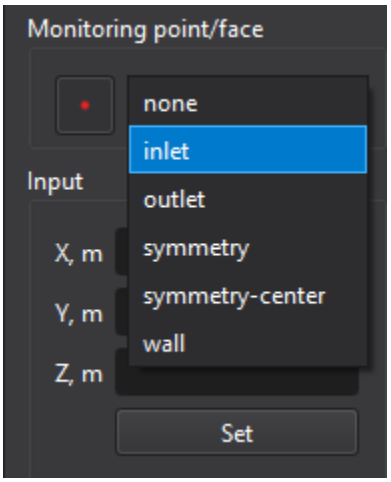


Figure 56. Monitoring point/face panel.

Monitoring entities can be deleted by pressing the RMB on a monitoring entity in the **Monitoring entities tree** and selecting **Delete** (Figure 57). Monitoring entities may also be hidden or unhidden by pressing the check mark box.

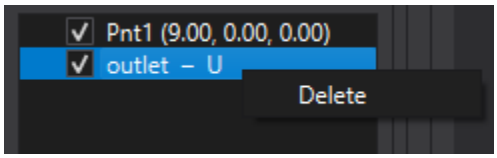


Figure 57. Deleting monitoring entities.

The monitoring entities are set in the Coordinates panel. Point can be defined via coordinate input, while face can be chosen from the **Selection** menu using their assigned names. By clicking on element in **Monitoring** entities tree the variable at point or face can be set (Figure 58).

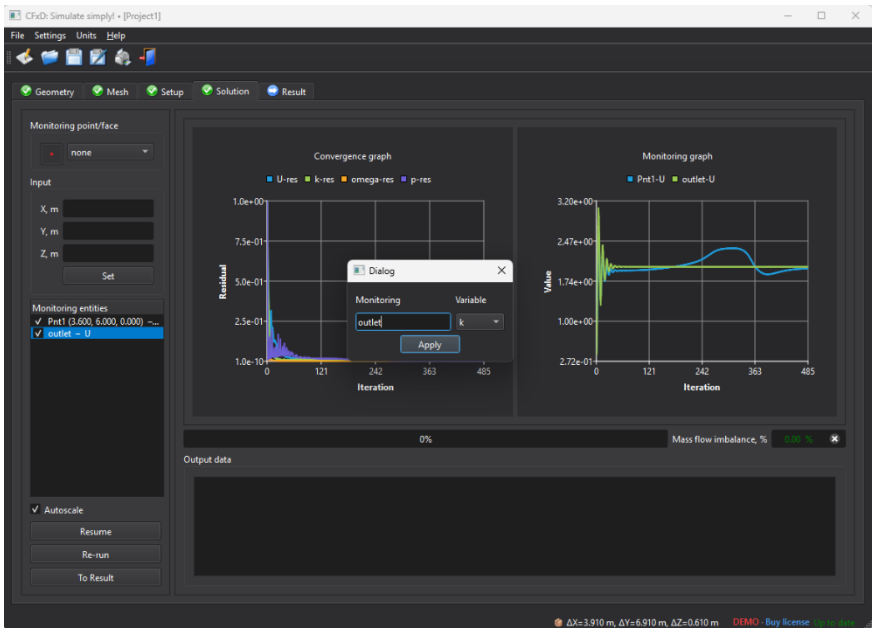


Figure 58. Setup of monitoring entity.

Convergence and monitoring curves can be viewed in the graphs on the right side of the program. The convergence graph has two display modes: **Linear** (default) and **Log scale**. Both graphs support **Autoscale**. When **Autoscale** is disabled, the graphs can be zoomed in and out manually. Below the graphs, the **Progress bar** and the **Mass flow imbalance** monitor are displayed. Detailed numerical values and important solver output messages are shown in the **Output data** window.

6.3 Start, Pause, Resume

Simulation execution is controlled using two buttons. Initially, only **Run** is enabled (Figure 59).

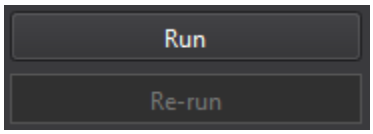


Figure 59. Initial buttons state.

After the simulation is started by clicking **Run**, the **Run** button changes to **Pause**, allowing the simulation to be paused (Figure 60).

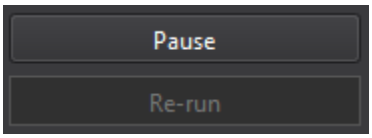


Figure 60. Button state after starting the run.

After clicking **Pause**, the second button becomes active, and the buttons change to **Resume** and **Re-run** (Figure 61). Clicking **Resume** continues the simulation from the paused state.

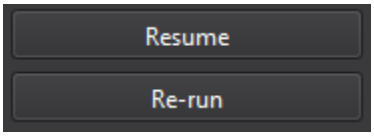


Figure 61. Button state after pausing.

The **Re-run** button restarts the simulation from the beginning. After the **Solution** step is completed, the next step, **Result**, can be opened using the **To Result** button.

Chapter VII. Result

The simulation results can be visualized in **Result** tab (Figure 62).

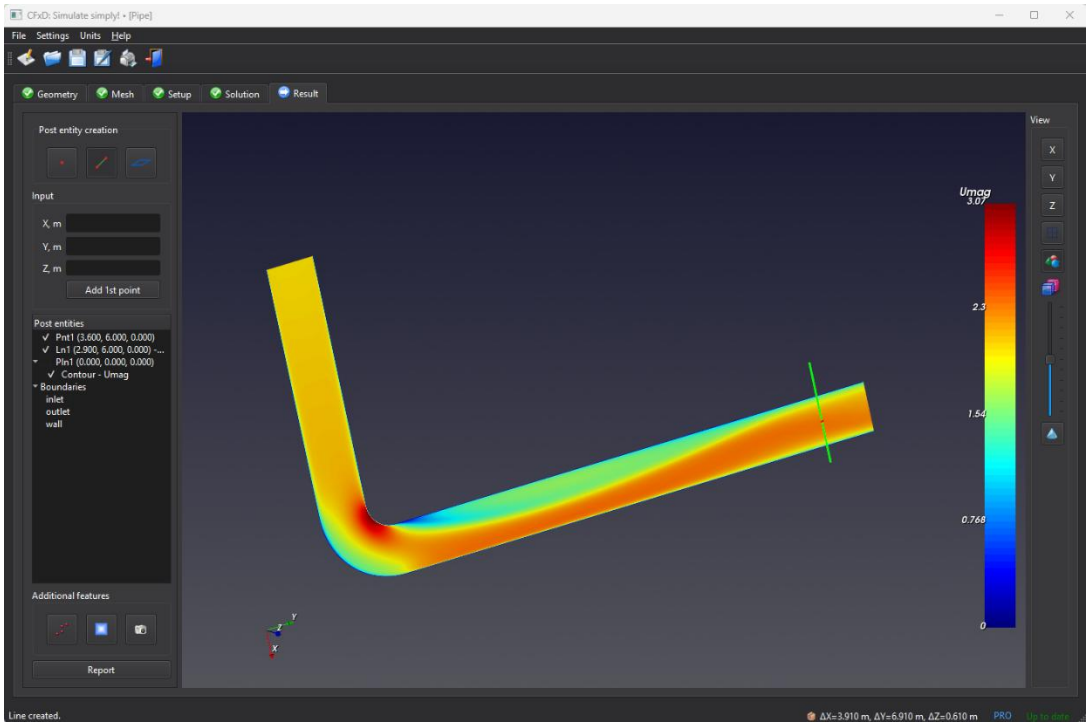


Figure 62. Result tab.

The results can be post-processed at a point (single value), along a line (chart) and on a plane (contour/vector).

7.1 Postprocessing at a point

The value of any variable at a point can be viewed in the dialog opened from the **Post entities** panel by clicking a point in the tree (Figure 63).

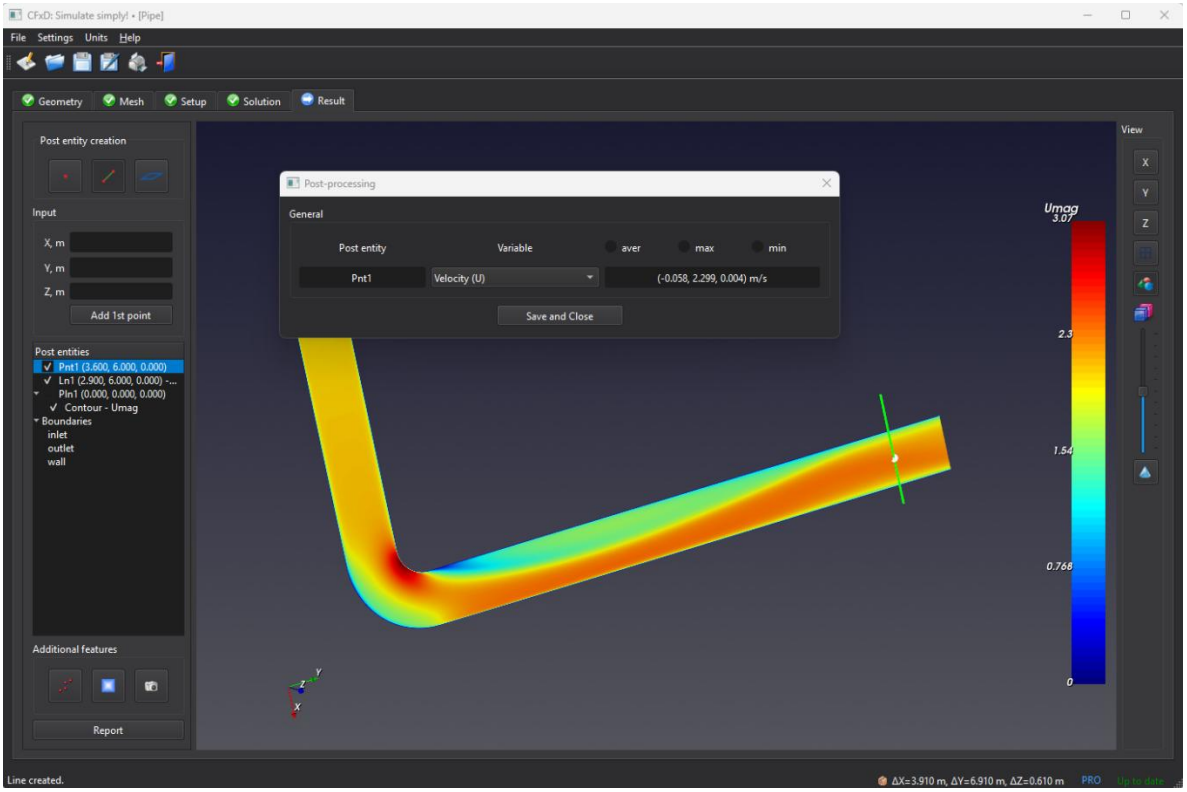


Figure 63. Velocity components value at a point.

7.2 Postprocessing along a line

Similarly, the values of any variable along a line (Chart) can be viewed in the dialog opened from the **Post entities** panel by clicking a line in the tree (Figure 64). In addition, the **average**, **minimum**, and **maximum** values along the line can be extracted. For comparison purposes, **external reference data** (e.g., experimental measurements or results from other simulations) in .csv format can also be added to the same chart. This allows CFxD results to be compared directly against a baseline dataset within a single plot.

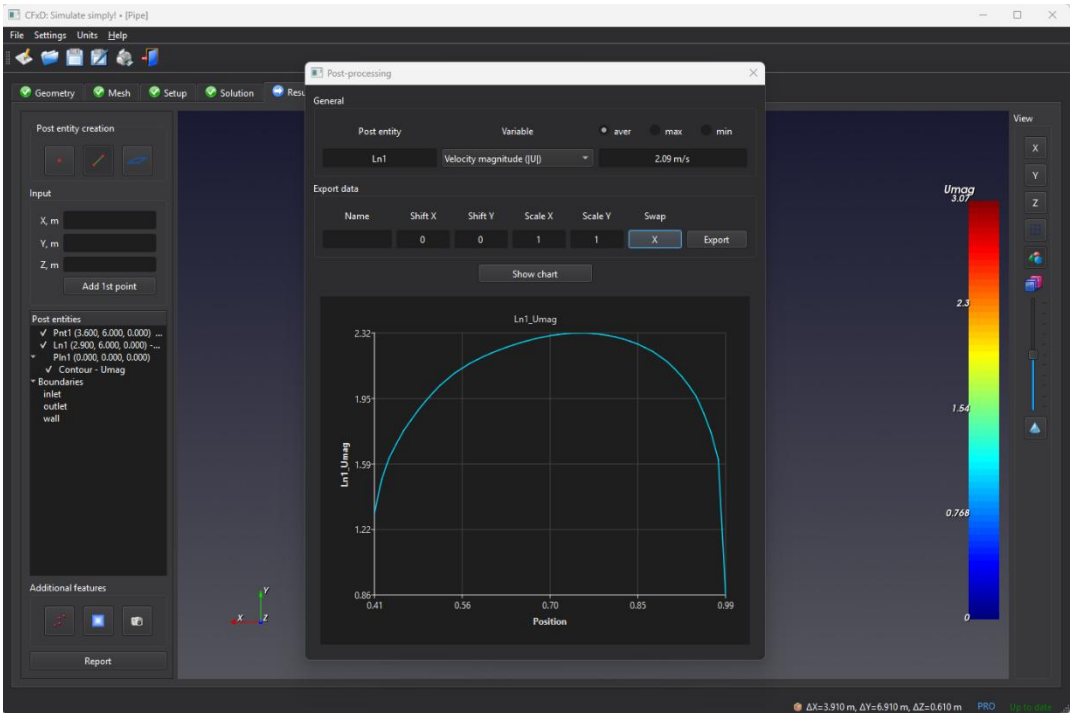


Figure 64. Chart at line.

7.3 Postprocessing on plane/boundary

The contour or vector plot can be visualized on a plane. It can be configured in the dialog opened from the **Post entities** panel by clicking a plane in the tree. The contour or vector plot is displayed in the **Graphics** window (Figure 65).

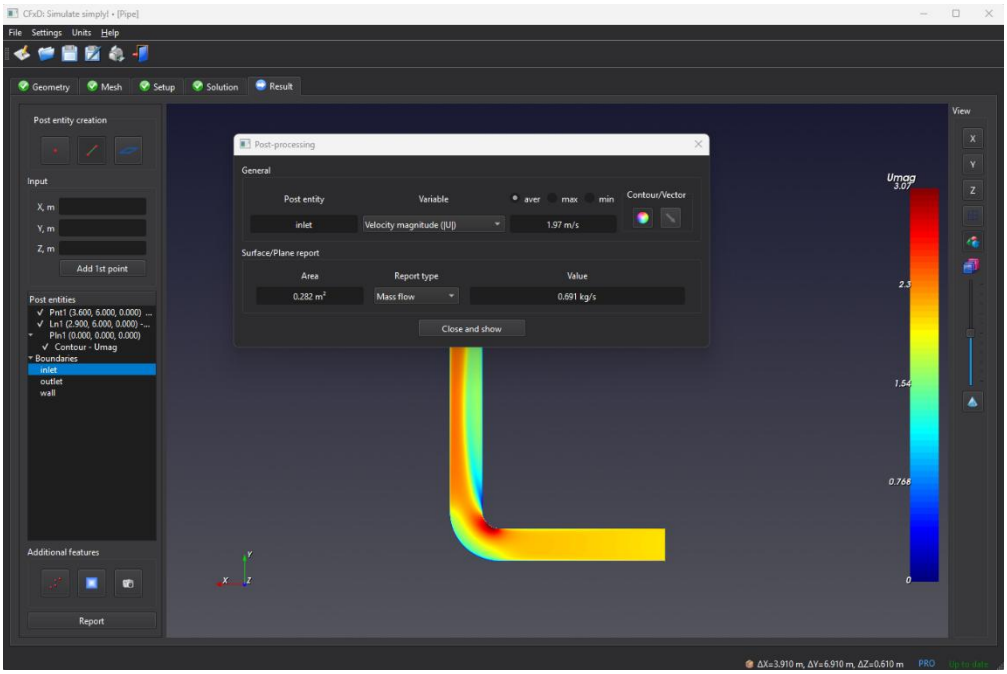


Figure 65. Velocity contour plot.

When **Velocity** is selected, the **Vector Settings** panel is enabled (Figure 66). It allows the vector display to be customized, including:

- **Vector scale**: set a manual scale or use **Auto** scaling.
- **Vector density**: increase or decrease the number of displayed arrows.
- **Uniform length**: display arrows with a constant length (direction only) or use auto magnitude-scaled arrows.

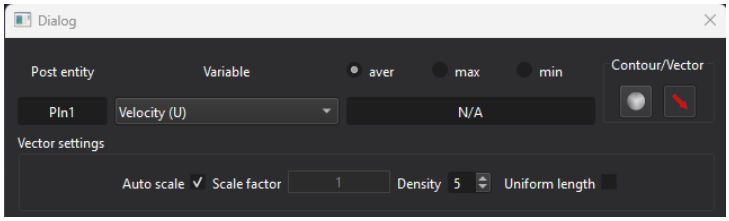


Figure 66. Vector settings panel.

The color bar on the right can be customized with a left or right mouse click. The minimum and maximum limits, as well as the number of ticks, can be adjusted (Figure 67). The color bar can also be reset to the default automatic settings.

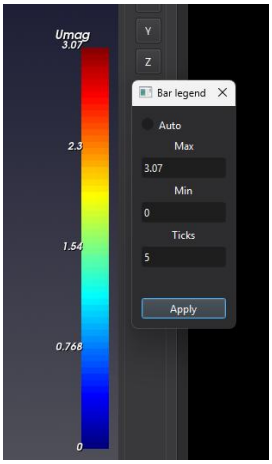


Figure 67. Customizing color bar.

Similar to line reports, average, maximum and minimum values can be extracted. In addition, mass flow rate, volumetric flow rate, and wall pressure force can be calculated.

7.4 Deleting and hiding post entities

Post entities can be deleted by pressing RMB over desired post entity in the “post entities window” and selecting delete (Figure 68). Monitoring entities may also be hidden or unhidden by pressing the check mark box.

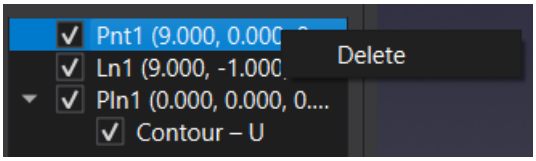


Figure 68. Deleting post entities.

7.5 Streamlines and isosurfaces

Streamlines can be visualized using the Streamlines button. The seeding surface (inlets/outlets) and number of streamlines can be specified in the Streamlines dialog (Figure 69). Streamline animation can also be enabled in the same dialog.

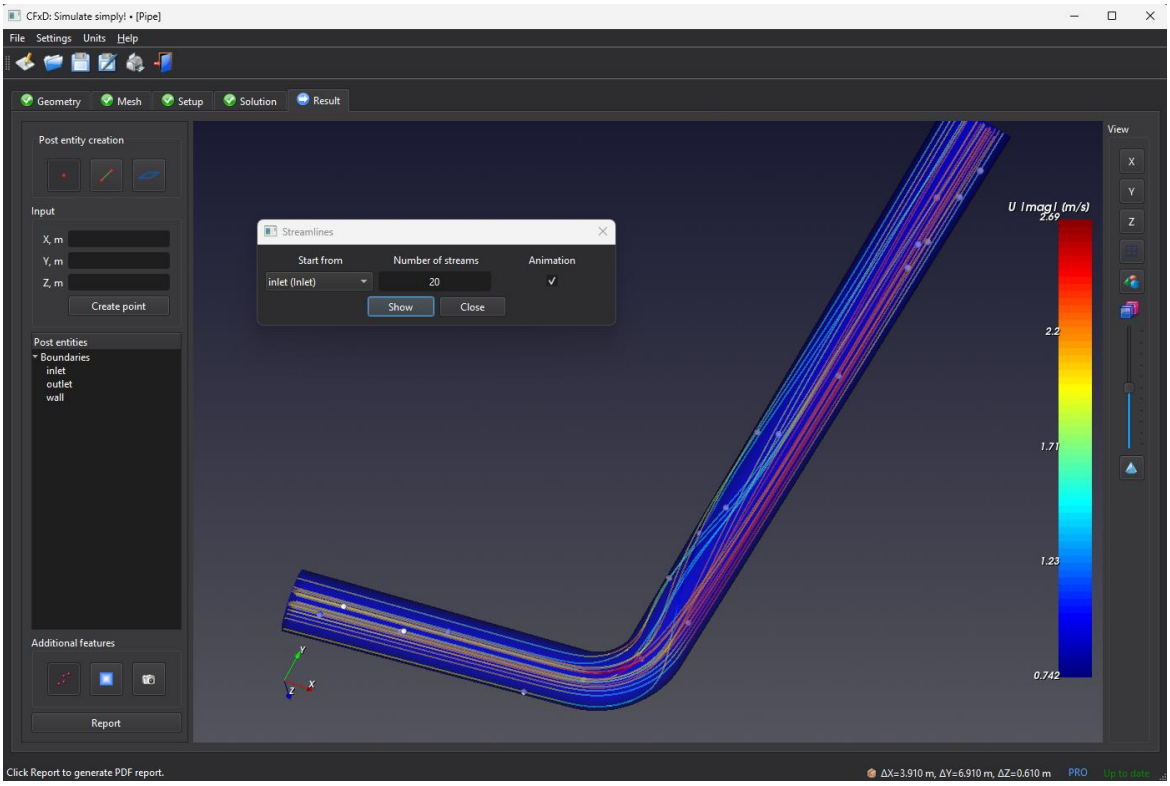


Figure 69. Streamlines setup.

Clicking the Isosurface button opens the Isosurface dialog, where the variable, isosurface value, and opacity can be specified (Figure 70).

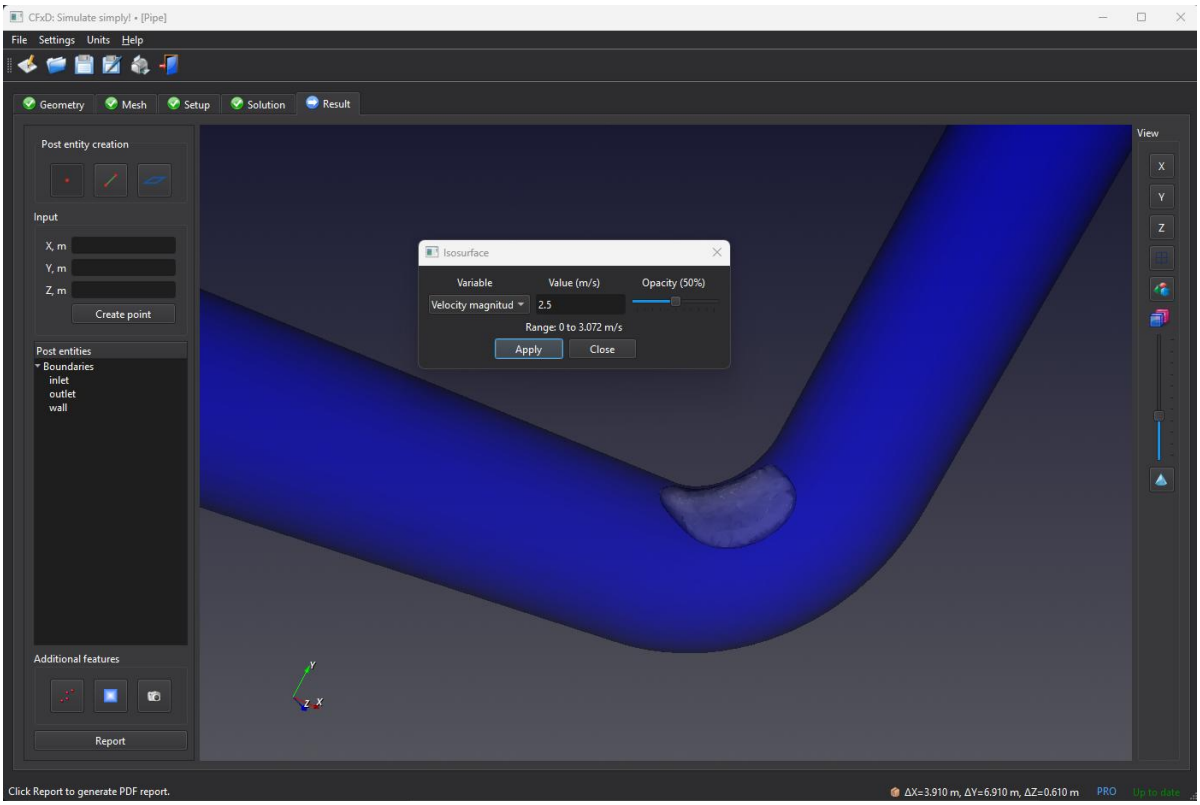


Figure 70. Isosurface setup.

7.6 Transient simulation result

For transient simulations, the **Post transient** panel is displayed. It allows results to be visualized at a selected time step. In addition, an animation dialog can be opened to create and play an animation (Figure 71). The animation is saved in the **Result** folder.

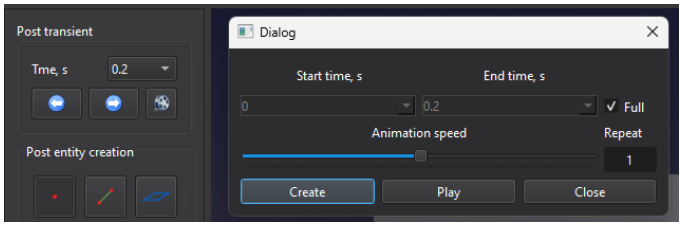


Figure 71. Post-processing for the transient simulations.

7.7 Report

The generated images can be saved to files. All important data can be compiled to PDF document using the **“Report”** button. An example of a generated report is shown in Figure 72.



CFxD Simulation Report

Project: Project1

Path: C:\CFxD\Projects\Project1

Date: 2025-12-07 12:53:31

Geometry

Statistics

- **Solids:** 1
- **Faces:** 3
- **Edges:** 3
- **Vertices:** 5

Mesh

Statistics

- **Cells:** 146204
- **Nodes:** 150861

Quality

- **Orthogonality (≥ threshold):** 94 %
- **Skewness (≤ threshold):** 40 %

Setup

Material properties

- **Material name:** Air
- **Density, ρ:** 1.225 kg/m^3
- **Kinematic viscosity, ν:** 1.81e-05 m^2/s

Run type

- **Mode:** Steady
- **Number of iterations, N:** 100
- **Write interval:** 10

Figure 72. Example of report.

References

1. Ferziger J. H., Peric M. Computational Methods for Fluid Dynamics. 3rd edition. Springer. 2002.
2. Schneider P. J., Eberly D. H. Geometric Tools for computer graphics. Elsevier Science. 2003.
3. Versteeg H. K., Malalasekera W. An introduction to Computational Fluid Dynamics.
4. VTK. Manual.
5. Open Cascade. Manual.
6. OpenFOAM. Manual.
7. OpenGL. Manual.
8. Qt. Manual.
9. Yun A. Computational Fluid Dynamics: from zero to guru. Creative Space, 2019.