

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

# InsightCAE Manual

November 24, 2021

## Contents

<b>1 Introduction</b>	<b>3</b>
1.1 License and Copyright . . . . .	3
1.2 About InsightCAE . . . . .	3
1.3 Contributors . . . . .	4
1.4 Features and Highlights . . . . .	4
1.5 Reading this Manual . . . . .	5
1.6 Recommendations for Working with Shell-based Tools in Linux . . . . .	5
<b>2 Additional Sources of Information</b>	<b>6</b>
2.1 In the Web . . . . .	6
2.2 Reporting Bugs and Feature Requests . . . . .	6
2.3 Getting Professional Help and Support . . . . .	6
<b>3 Obtaining InsightCAE</b>	<b>7</b>
3.1 Installation using Linux Package Manager . . . . .	7
3.2 Installation on Windows . . . . .	7
Usage . . . . .	7
3.3 Building from Sources . . . . .	8
3.3.1 Getting the Source Code . . . . .	8
3.3.2 Dependencies . . . . .	8
3.3.3 Building . . . . .	8
3.4 Setting up the Environment . . . . .	9
<b>4 Overview of the Project InsightCAE</b>	<b>9</b>
4.1 Concept and Parts of the Project . . . . .	9
4.2 Terms and Definitions . . . . .	9
Simulation App, Analysis . . . . .	9
Parameter Set . . . . .	10
Result Set . . . . .	10
Case Element . . . . .	10

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

<b>5 Simulation Apps</b>	<b>11</b>
5.1 Explanation of Building Bricks . . . . .	11
5.1.1 Input: Parameter Sets . . . . .	11
5.1.2 Output: Results Sets . . . . .	11
5.1.3 Analyses: the Simulation Procedure . . . . .	11
Python Script . . . . .	12
C++ . . . . .	13
5.2 Usage . . . . .	13
5.2.1 GUI: workbench . . . . .	13
Editing the Input Parameters . . . . .	13
Saving Parameters . . . . .	14
Running the Analysis . . . . .	14
Cancel a Simulation . . . . .	15
Viewing the Results . . . . .	15
Rendering Results into a Report . . . . .	15
Modifying a Parameter . . . . .	15
Creating a New Analysis . . . . .	15
5.2.2 Special Features for OpenFOAM-based Simulation Apps . . . . .	15
Restart Behavior . . . . .	16
Launching Graphical Postprocessor ParaView . . . . .	16
5.2.3 Console on Headless Computers: analyze . . . . .	16
5.3 Available Simulation Workflows . . . . .	16
5.3.1 Numerical Wind Tunnel . . . . .	16
5.3.2 Internal Pressure Loss . . . . .	16
<b>6 Supporting Simulations with OpenFOAM</b>	<b>19</b>
6.1 Case Setup GUI: isofCaseBuilder . . . . .	19
6.2 Execution Monitor GUI: isofExecutionManager . . . . .	19
6.3 Tabular Output Data Plot: isofPlotTabular . . . . .	19
6.4 Common Tasks in the Context of OpenFOAM Simulations . . . . .	19
6.4.1 Controlling and Checking Running OpenFOAM Case Which Were Generated by InsightCAE . . . . .	19
Trigger Output Independently from the Configured Output Interval . . . . .	19
Trigger Immediate Output and Stop Solver Afterwards Without Raising an Error . . . . .	19
Inspecting a Running Case Using Paraview . . . . .	19
6.4.2 Meshing . . . . .	21
Extraction of Feature Edges from STL Geometries . . . . .	21
<b>7 Extensions to Back-End-Tools</b>	<b>28</b>
7.1 OpenFOAM . . . . .	28
<b>8 InsightCAD</b>	<b>28</b>
8.1 Introduction . . . . .	28
8.2 Basic Concept . . . . .	28
General CAD Model Script Syntax . . . . .	29
8.3 Submodels and Subassemblies . . . . .	29
8.4 Symbol Definition . . . . .	29
8.4.1 Scalars . . . . .	29
8.4.2 Vectors . . . . .	30
8.4.3 Datums . . . . .	31

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

8.4.4 Features . . . . .	32
8.5 Feature Commands . . . . .	32
8.5.1 Transformation . . . . .	33
8.5.2 Import . . . . .	33
8.5.3 Geometry Construction . . . . .	33
Primitives . . . . .	33
Other Feature Commands . . . . .	34
8.6 Lower Dimensional Shape Selection . . . . .	34
8.6.1 Vertices . . . . .	35
8.6.2 Edges . . . . .	35
8.6.3 Faces . . . . .	36
8.6.4 Solids . . . . .	36
8.7 Postprocessing Actions . . . . .	37
8.7.1 Drawing Export . . . . .	37
8.7.2 Mesh Creation . . . . .	37
8.8 Graphical Editor ISCAD . . . . .	39
8.8.1 3D Graphics Display . . . . .	40
View Manipulation . . . . .	40
Model Navigation . . . . .	40
8.8.2 Text Editor . . . . .	42
<b>9 Developer Documentation</b>	<b>43</b>
9.1 Plug-In Development . . . . .	44
9.2 InsightCAE Development . . . . .	44
9.2.1 Dependencies . . . . .	44
9.2.2 OpenFOAM Analysis . . . . .	44
<b>10 Getting Started, Tutorials</b>	<b>45</b>

## 1 Introduction

### 1.1 License and Copyright

InsightCAE is free software; you can redistribute it and/or modify it under the terms of version 2 of the GNU General Public License<sup>1</sup> as published by the Free Software Foundation. You accept the terms of this license by distributing or using this software.

This manual is Copyright (c) 2017-2021 silentdynamics GmbH.

Permission is granted to copy, distribute and/or modify this document under the terms of the GNU Free Documentation License, Version 1.3 or any later version published by the Free Software Foundation; with no Invariant Sections, no Front-Cover Texts, and no Back-Cover Texts. A copy of the license is included in the section entitled "GNU Free Documentation License".

---

<sup>1</sup><http://www.gnu.org/licenses/old-licenses/gpl-2.0.html>

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

## 1.2 About InsightCAE

InsightCAE is a framework for building automated analysis and design workflows in Computer-Aided Engineering with a preference for open source tools.

Individual open source projects often meet only subtasks in an analysis process and have very different APIs. To conduct computational tasks, a combination of several software tools is often required. In order to use open source software productive and efficient for everyday tasks, the analysis automation framework InsightCAE was created.

InsightCAE serves as a framework for the implementation of analysis procedures. The objective is to provide adapters and interfaces to the tools and simulation programs that are needed for a specific computing task.

Some common use cases are:

- create a simulation app

InsightCAE's toolkit library is used to build a simulation app. The workbench or the web interface can be used to present the parameter form to the user and display a 3D preview of the setup. The app generates a result set. InsightCAE contains a viewer to display result sets or compare data from different result sets. The simulation app can be stored in a custom library or in a python script.

- assist in creation of OpenFOAM cases

InsightCAE contains a GUI (called "Case Builder") to build OpenFOAM cases by putting features together into an OpenFOAM case.

- build parametric, script-based CAD models

InsightCAE contains an interpreter for script-based CAD models. The underlying CAD kernel is OpenCASCADE (a BREP kernel). This can be utilized to produce geometry in the course of optimizations, for example.

## 1.3 Contributors

The following people have so far contributed to InsightCAE:

- Hannes Kröger (silentdynamics GmbH)
- Johann Turnow (silentdynamics GmbH)

If you want to get involved in the development, please feel invited to do so! We greatly appreciate any contribution and we would very much like to add you to the above list.

If you made some modification or addition to the code, which you would like to be merged into the main development line, please consider to send us a pull request<sup>2</sup>.

## 1.4 Features and Highlights

InsightCAE's objective is to create automated analysis workflows. The high level API resides in the core "toolkit" library. Automated workflows usually involve different external programs and

<sup>2</sup><https://help.github.com/en/github/collaborating-with-issues-and-pull-requests/about-pull-requests>

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

utilities. For the realization of automated workflows, it is sometimes required, to create add-ons to these external programs. Thus InsightCAE is also a container for add-ons to other programs.

- InsightCAD script-based, fully parametric CAD
  - OpenCASCADE geometry kernel
  - assemblies, constraint-based sketches, part library, drawing export
- OpenFOAM add-ons (schemes, boundary conditions, models, ...)
- Pre- & Postprocessing tool (OpenFOAM Case Builder)
- analysis workflow automation tools (GUI)

## 1.5 Reading this Manual

## 1.6 Recommendations for Working with Shell-based Tools in Linux

Sometimes it is necessary to execute tools in a bash shell, e.g. because no GUI for it is available. And often it is easier to keep an overview in graphical file manager. Having a console window open together with a file manager window at the same time is an obvious solution but to do so for many cases at a time may easily confuse the desktop.

Here are two useful hints to solve this issue:

1. The KDE file manager **dolphin** offers the possibility to display a console in the lower half of the window (figure 1). The working directory is synchronized with the directory shown in the graphical window by injection of **cd** commands. Vice versa, when the working directory is changed in the shell, the graphical display is updated as well.
2. There is a Norton-Commander-like file manager named **krusader** (figure 2). It offers the same functionality regarding the embedded terminal but a more flexible way of displaying multiple folders. In addition to the two list view on the left and right, multiple tabs for folders can be added in each list view. And there is also a rich interface to define custom commands and file associations.

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

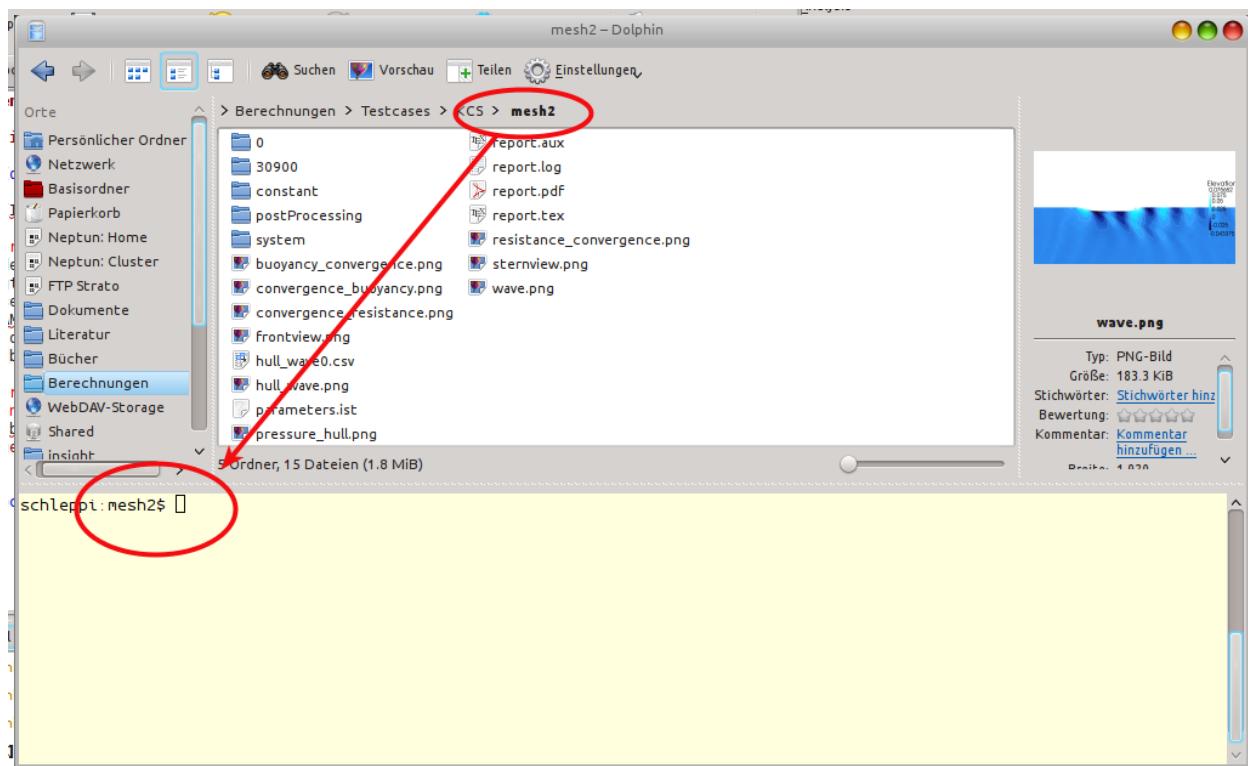


Figure 1: Dolphin file manager with embedded terminal

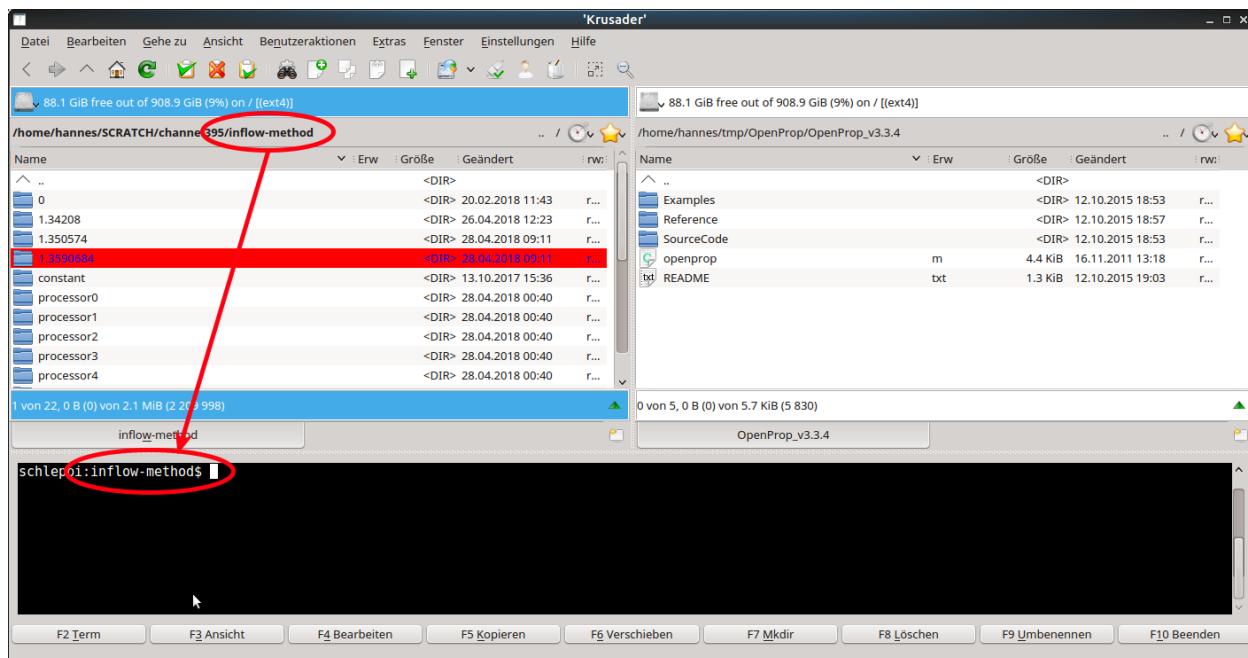


Figure 2: Krusader file manager with embedded terminal

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

## 2 Additional Sources of Information

### 2.1 In the Web

- Source Code Repository <https://github.com/hkroeger/insightcae>
- Issue Tracker <https://github.com/hkroeger/insightcae/issues>
- Web Forum <https://groups.google.com/forum/#!forum/insightcae>

### 2.2 Reporting Bugs and Feature Requests

Please use the issue tracker <https://github.com/hkroeger/insightcae/issues> to report any bugs.

We also monitor the web forum <https://groups.google.com/forum/#!forum/insightcae> for questions or feature requests.

### 2.3 Getting Professional Help and Support

Beyond the web resources above, silentdynamics GmbH offers commercial support<sup>3</sup> for professional users of InsightCAE. Automated analyses according to customer needs and specifications are implemented by creating new specialized modules for Insight CAE. Typical support contracts include also user training and continuous customization and updating of InsightCAE and its add-ons.

## 3 Obtaining InsightCAE

### 3.1 Installation using Linux Package Manager

We provide binary installation packages for Ubuntu-based Linux distributions. We specifically support the latest Ubuntu LTS distribution.

The packages are held in our repository. To install them, first add the silentdynamics apt repository and the associated key by executing:

```

1 $ sudo apt-key adv --fetch-keys http://downloads.silentdynamics.de/
  SD_REPOSITORIES_PUBLIC_KEY.gpg
2 $ sudo add-apt-repository http://downloads.silentdynamics.de/ubuntu
3 $ sudo apt-get update

```

Then, install the software by executing

```
1 $ sudo apt-get install insightcae-ce
```

Continue with preparing the environment according to section 3.4

---

<sup>3</sup><http://silentdynamics.de/en/oss-cae/>

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

## 3.2 Installation on Windows

Our software as well as OpenFOAM and Code\_Aster are currently pure Linux software. However, it can also be run under Windows 10 (64 bit only) using the "Linux Subsystem for Windows". Virtualization is not used. The processes share the memory with the Windows processes and the files are stored in the Windows file system. For graphical output under Windows, a X server has to be installed. Various X servers are available.

For the comfortable installation of InsightCAE and OpenFOAM under Windows we provide an installer:

[https://sourceforge.net/projects/insightcae/files/install\\_InsightCAE.exe](https://sourceforge.net/projects/insightcae/files/install_InsightCAE.exe)

Download the EXE file from this link and execute it:

- It will activate the Windows subsystem for Linux, if it is not already activated
- Then Ubuntu 18.04 is installed in the WSL environment
- Afterwards, the current version of InsightCAE and OpenFOAM is downloaded from the Internet and installed in the WSL environment. For this purpose, about 500MB are transferred over the Internet and about 3GB of hard disk space is required.
- Finally the X-Server vcXsrv is installed.
- Please note:
  - For the activation of WSL and the installation of vcXsrv administrator rights are required
  - The WSL system is set up for the user running the installer. Other users of the system cannot use the installation.

**Usage** First launch "Xming" X-Server.

Then start Ubuntu Bash command line. Execute the application in the shell, e.g. the InsightCAE Workbench:

```
1 $ workbench
```

## 3.3 Building from Sources

### 3.3.1 Getting the Source Code

The source code of InsightCAE is hosted at Github. Cloning a working copy constitutes the best way to get updates and the latest bug fixes.

Alternatively, you can download snapshot archives from github, if you don't want to bother with a git client. In this case, just replace the cloning step below with unpacking the archive.

First, get the the sources by cloning the git repository:

```
1 $ git clone https://github.com/hkroeger/insightcae.git insight-src
```

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

### 3.3.2 Dependencies

The InsightCAE project depends on a number of other projects. Some of the dependencies are optional and only required, if certain features are enabled. The dependencies are listed in table 1.

Project	tested version	required for
Armadillo	9.800.4	(always)
Gnu Scientific Library	2.6	(always)
Boost	1.65.1	(always)
VTK	9	(always)
Python (Lib)	3.6	(always)
OpenCASCADE	7.4	(always)
DXFlib	3.7.5	(always)
libpoppler	0.86	(always)
Qt	5.15	for GUIs
Gmsh	4.8	only for analyses with tet/tri meshing
Wt toolkit	4.1	only for client/server execution
OpenFOAM	ESI1806, 4.1 extend	only for OpenFOAM based analyses
Code_Aster	14	only for FEM analyses
SWIG	4	only for python bindings

Table 1: Required dependencies for building InsightCAE

### 3.3.3 Building

CMake is utilized for managing the build. The preferred way is to build the software out of source in a separate build directory. Create a build directory, then configure the build using e.g. `ccmake` and finally build using `make`:

```
1 $ mkdir insight && cd insight
2 $ ccmake ../insight-src
```

In the CMake GUI, you may need to change paths or adapt settings. Please refer to the CMake documentation<sup>4</sup> on how to use CMake. The biggest difficulty will probably be, to provide all the software packages, on which InsightCAE depends.

The final step in CMake is, to generate the Makefiles. Once this is done without errors, start the compilation of the project by executing:

```
1 $ make
```

To work with InsightCAE, some environment variables are needed to be set up. A script is provided therefore. It can be parsed e.g. in your `~/.bashrc` script by adding to the end:

```
1 $ source /path/to/insight/bin/insight_setenv.sh
```

<sup>4</sup><https://cmake.org/cmake/help/latest/guide/tutorial/index.html>

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

### 3.4 Setting up the Environment

To set up the environment variables and search paths required by the InsightCAE components, a script needs to be parsed. This is conveniently added to the shell startup script file `~/.bashrc`. Add the following line to the *beginning* of the `~/.bashrc` file:

```
1 source /opt/insightcae/bin/insight_setenv.sh
```

Note that the default contents of this file differs between linux distributions. Sometimes it contains statements which prematurely end its evaluation for non-interactive shells. This

## 4 Overview of the Project InsightCAE

### 4.1 Concept and Parts of the Project

The InsightCAE builds on top of available open-source engineering analysis tools (though closed-source tools could be used as well). The most used tool by far is the CFD software OpenFOAM.

For certain tasks, custom extensions to these tools are required. InsightCAE contains a selection of extensions, which are included in the distribution packages, when these are created.

The core of the project is the toolkit library. It contains the implementation of core classes in workflow management as the parameter set storage and tools and result set storage classes and handling tools. The toolkit library also maintains a list of available analysis modules, which is dynamically extensible by loadable add-ons.

Finally, on top of the toolkit library, custom analysis modules can be created. The InsightCAE base distribution contains a number of generic analysis modules as the "Numerical Wind Tunnel" module and some generic test case analyses like flat plate, channel flow, pipe flow and so on. These might serve as examples for custom implementations.

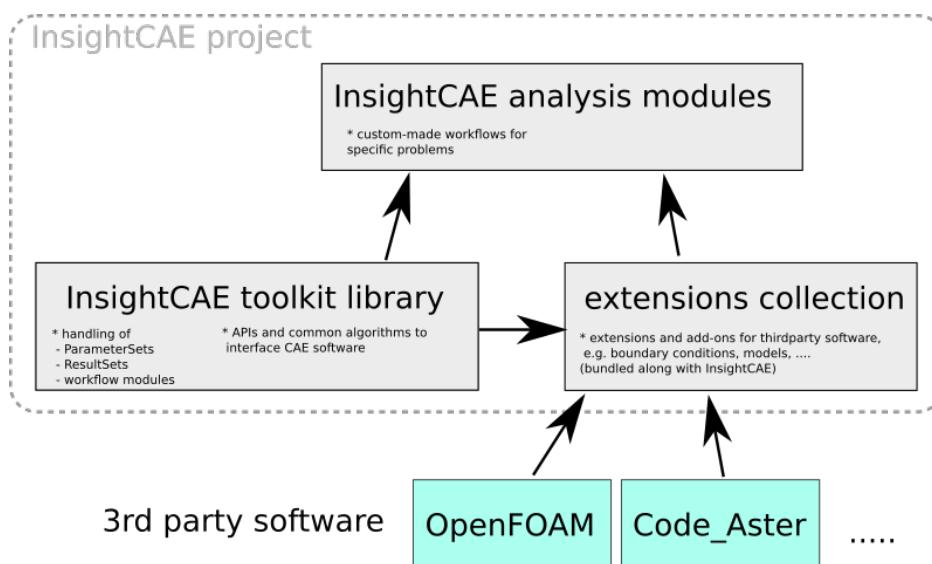


Figure 3: Structure of the InsightCAE project

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

## 4.2 Terms and Definitions

**Simulation App, Analysis** InsightCAE bundles tools to create automated simulation procedures. These procedures can be written in Python or C++. The term **analysis, workflow module** or **Simulation App** is used for such a procedure.

Actually, an analysis is like a function in programming: it gets input parameters, executes some actions (e.g. a numeric simulation) and returns a results data structure.

InsightCAE provides a GUI to edit the parameter sets and also tools to work with the result sets. E.g. convert the result set into a PDF report.

**Parameter Set** The analysis modules need input data, i.e. parameters. All parameters for a module are grouped together in a parameter set. Parameter sets are stored in XML files in human readable format (extension \*.ist). They can also be edited in a GUI editor (application workbench).

Many analysis modules need external files (like CAD data in STL or any other format). These files can simply be referred in a parameter set by their relative path (relative to the \*.ist-file containing the reference) or absolute path. It is also possible to embed the external files into the XML file in an base64-encoded form. This simplifies transfer of analysis setup from one computer to another, e.g. for remote execution.

**Result Set** The result of an analysis is returned in a **result set**. This is a hierarchical data structure which contains nodes like scalar values, charts, contour plots, images, tables or similar entities. It can be stored on disk in XML format. There is also a renderer which converts a result set into a PDF report.

**Case Element** The setup of CFD-runs is put together from a number of so-called **case elements**. Each case element represents some functional feature. It might be a boundary condition or a turbulence model or something similar. A case element is not necessarily associated with a certain configuration file but can modify multiple configuration files of a CFD case configuration.

There is also an editor for composing OpenFOAM cases from the available case elements: the OpenFOAM Case Builder.

- Solver extensions
- OpenFOAM Case Builder
- Workbench

## 5 Simulation Apps

### 5.1 Explanation of Building Bricks

#### 5.1.1 Input: Parameter Sets

Parameter sets are a hierarchical collection of parameters. Parameters can be of different type, see table 2 for a list.

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

A parameter set constitutes the full input data set for an analysis.

Beyond these primitive parameter types, there can be compounds, namely:

- **subset**  
a subdirectory in a parameter set
- **selectablesubset**  
a subset that switches its content based on a **selection** parameter
- **array**  
an array of either a compound parameter or a primitive parameter

Type	Description
double	a scalar floating point value
int	an integer value
bool	a boolean value
vector	a 3-tuple of doubles
string	a simple text
selection	a selection from a predefined set
path	a file path, relative to the input file
matrix	a matrix with arbitrary number of rows and cols
doublerange	an array of double values

Table 2: Available primitive parameter types in a parameter set

### 5.1.2 Output: Results Sets

The output of an analysis is stored in a result set. It may contain elements like charts, images, tables, numbers etc. A result set can be saved to a XML file and/or rendered into a PDF report.

### 5.1.3 Analyses: the Simulation Procedure

The procedure of a simulation is contained in a so-called analyses or workflow module. You may think of it as an app for executing a certain type of simulation. It is essentially a program. Often times, people create scripts or macros for speeding up their simulation workflows. The InsightCAE workflow modules serve a similar purpose, though the intention is to fully cover the whole simulation including all pre and post processing steps.

There is a GUI available which provides an editor for the input parameter set (see section 5.2.1). The GUI is also capable of displaying a 3D preview of the involved geometry and other elements, which are displayable. The analysis module is responsible of creating the 3D preview elements. Creation of the preview is optional and not every analysis module produces a preview.

The GUI also includes a viewer for the result set. From that viewer, the reports may be exported.

InsightCAE holds a runtime-extensible list of available analyses. This list can be extended at run time either by providing shared libraries containing more analysis modules or by python scripts.

The analyses can be programmed essentially in C++ or in Python. InsightCAE is essentially written in C++. Python wrappers for the relevant functions and objects are created using SWIG. So the API

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

can also be accessed from Python.

If a new analysis shall be created, this parameter editing and result viewing infrastructure is easily reusable:

- The new analysis module just needs to provide the parameter set layout by returning the default parameter set (a parameter set filled with suitable default values),
- it has to provide a function for executing the analysis.
- This function needs to return a result set.

The analysis modules may group themselves into categories. This is used in the initial analysis creation dialog in the GUI.

**Python Script** A python based analysis comprises a single dedicated python script file. The InsightCAE functions are included by an initial statement like `from Insight.toolkit import *`.

This script needs to define three standalone functions:

1. `category()`

This functions returns a plain string with the category of the analysis. Multiple hierarchy levels are supported and have to be separated by slashes.

2. `defaultParameters()`

This function shall return an object of type `ParameterSet`, containing the default values of all the needed parameters. The user cannot add or delete parameters, just modify their values. So this defines the layout.

3. `executeAnalysis(parameters, workdir)`

This function finally runs the simulation. It gets an object of type `ParameterSet`, containing all the input parameters and the string `workdir`, which contains the execution directory.

Upon loading, the InsightCAE base library searches in

- `$INSIGHT_GLOBALSHAREDDIRS/python_modules` and
- `$INSIGHT_USERSHAREDDIRS/python_modules`

for files named `*.py`. Each of these files will be loaded and their defined analyses will become available in the GUI and all other InsightCAE tools.

**C++** In C++, a new analysis is derived from the common base class `Analysis`. The necessary functions have to be overridden. The new analysis should be put into a dedicated shared library.

Upon loading, the InsightCAE base library searches in

- `$INSIGHT_GLOBALSHAREDDIRS/modules.d` and
- `$INSIGHT_USERSHAREDDIRS/modules.d`

for files named `*.module`. Each of these files has to contain a statement `library <FILENAME>`, where `FILENAME` is the name of the shared library file. These libraries will all be loaded and their defined analyses will become available in the GUI and all other InsightCAE tools.

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

## 5.2 Usage

### 5.2.1 GUI: workbench

The GUI for editing analysis parameters, run analyses and monitor their progress and to review the result sets is called **workbench**.

It can be started from the command line by executing

```
1 $ workbench
```

or from the global application menu.

The workbench can open and handle multiple analyses at a time. For each analysis, a sub window is opened (analysis form).

An analysis can either be created from scratch (see “Creating a New Analysis”) or by opening an existing analysis input file (select **Analysis > Open...** in the menu).

The analysis form has three tabs. From left to right: the input parameter edit tab, the progress display tab and the result viewer tab.

**Editing the Input Parameters** The parameter input tab is shown in figure 5. On the left, there is a tree view showing all the available parameters of the analysis. The font display style of the parameter entry denotes the importance of the parameter:

- highlighted yellow background: these parameters have no reasonable default values and need to be changed for every case. The displayed default values are only chosen to demonstrate some functional value, e.g. for speeds or rpms. But for parameters like file names, for example, there is no reasonable default at all.
- normal black text color: these parameters have reasonable default values, with which an analysis can be started. Though it is quite normal, that these parameters might need to be adapted from case to case.
- dimmed gray text color: these parameters should normally be left at their default values. But it for rather rare special cases, it might become necessary to adapt them.

The parameters can be selected my mouse click and edited (see “Modifying a Parameter”).

Some analyses (this is optional) support a 3D preview of the configured case. When an analysis is loaded, which supports 3D preview, a 3D view and a 3D element tree appears right of the parameter edit column. All views in the Input tab are shown in a horizontal splitter and their width can be adapted by dragging the splitter handle between them. It is also possible to hide the 3D preview widgets by collapsing their width to zero using the splitter handle (This might also occur accidentally). The collapsing can be undone by locating the splitter handle of the collapsed widget and dragging it back to some non-zero width. The layout of the splitter is saved on program exit and restored on the next start.

**Saving Parameters** Once all parameters are edited, the parameter set can be saved to a file. Very often, external files are referenced from a parameter set. Since it is also often necessary, to relocate parameter sets from one computer to another or between directories, InsightCAE parameter sets provide some special features which shall simplify relocation of parameter sets:

- file paths are always stored as relative paths inside the parameter set file.

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

Even if the workbench editor displays an absolute path, the paths are made relative to the saving location of the parameter set file. It is still possible to insert absolute paths into a parameter set file. But when it is loaded and saved again, these will be made relative.

If a parameter set is moved around together with the files it references, they will be found, as long as they remain in the same relative location, e.g. side by side with the parameter file or in a sub directory.

- external files can be embedded into the parameter file.

This is the preferred option, when parameter files shall be moved across computers. Upon saving, the files will be base-64 encoded (not compressed) and stored in the XML file. When the analysis is run, the files are extracted to a temporary location. The extraction is aware of the case, that different files with the same name might exist in different directories and extracts them to different locations. By default, the files are extracted to the system temporary location. When the InsightCAE application, which triggered the extraction of the parameter, exits, the files are deleted again from the temporary location. Note that this might not be achieved, if the application crashes.

Packing of external files can be enabled by checking or by checking the appropriate option in the "Save Parameters" dialog. Once, a parameter file was saved as packed or unpacked, the selection state is saved and used for all subsequent save operations.

**Running the Analysis** With a properly edited parameter set, the analysis run can be started. This is achieved by clicking on the button "Run" on the right side or by selecting in the menu. This analysis form switches automatically to the "Run" tab (see figure 6). In the lower half, the log message are displayed. Sometimes, errors in an external program occur and are not correctly reported. It is therefore a good practice to watch the log messages for errors or warnings.

When the simulation solver runs, InsightCAE shows progress variables graphically. The plots are shown above the log widget. The number and meaning of the progress variables depend on the analysis conducted.

Especially for OpenFOAM-bases analyses, a number of quantities are extracted from solver output and forwarded to the GUI as progress variables:

- continuity errors
- equation residuals
- forces
- moments
- execution time (per timestep)

**Cancel a Simulation** To cancel a simulation run, click on the button "Kill" on the right side or by select in the menu.

**Viewing the Results** Once the simulation run is finished (that means that the solver has finished and that all post processing steps are done), a result set is created and loaded into the workbench. It is displayed in the "output" tab (see figure 7). The result set can be rendered into a report (see "Rendering Results into a Report").

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

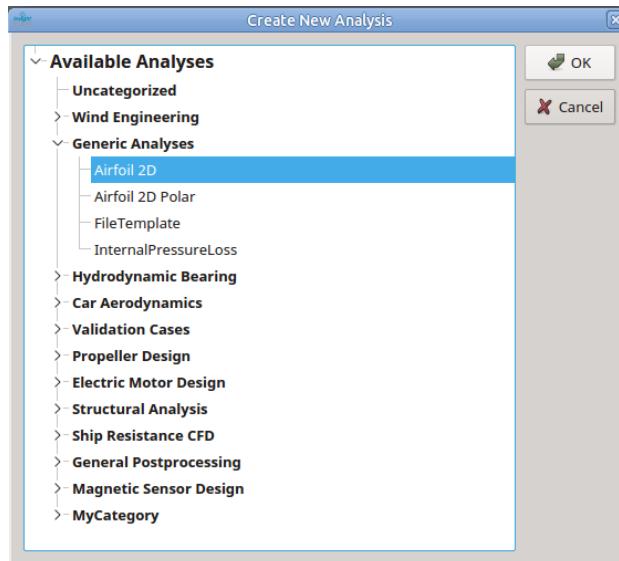


Figure 4: Dialog for selection of the type of a new analysis

## Rendering Results into a Report

### Modifying a Parameter

**Creating a New Analysis** A new analysis is created by selecting in the menu **Analysis** **New...**. Then a new dialog appears, in which the list of available analyses is displayed (figure 4). The required analysis should be selected and confirmed by clicking "Ok".

### 5.2.2 Special Features for OpenFOAM-based Simulation Apps

Simulation apps which are based on an OpenFOAM simulation provide some special features which is documented in this section.

**Restart Behavior** Since OpenFOAM-based simulations often run for a long time. It frequently appears that the procedure is interrupted and needs to be restarted.

There is not explicit function for a restart. It happens implicitly, when the analysis is re-launched in an existing working directory.

Depending on the state of the data, the behaviour will be as follows:

- the mesh exists (in folder "constant/polyMesh/") and a time directory exists (e.g. "0/"): it is assumed that the case is ready for execution and the solver will be restarted,
- only the mesh exists ("constant/polyMesh/") and no time directory: mesh creation will be skipped but the dictionaries will be recreated.

Depending on where the interruption happened, the state might be inconsistent and invalid behavior might result. For example:

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

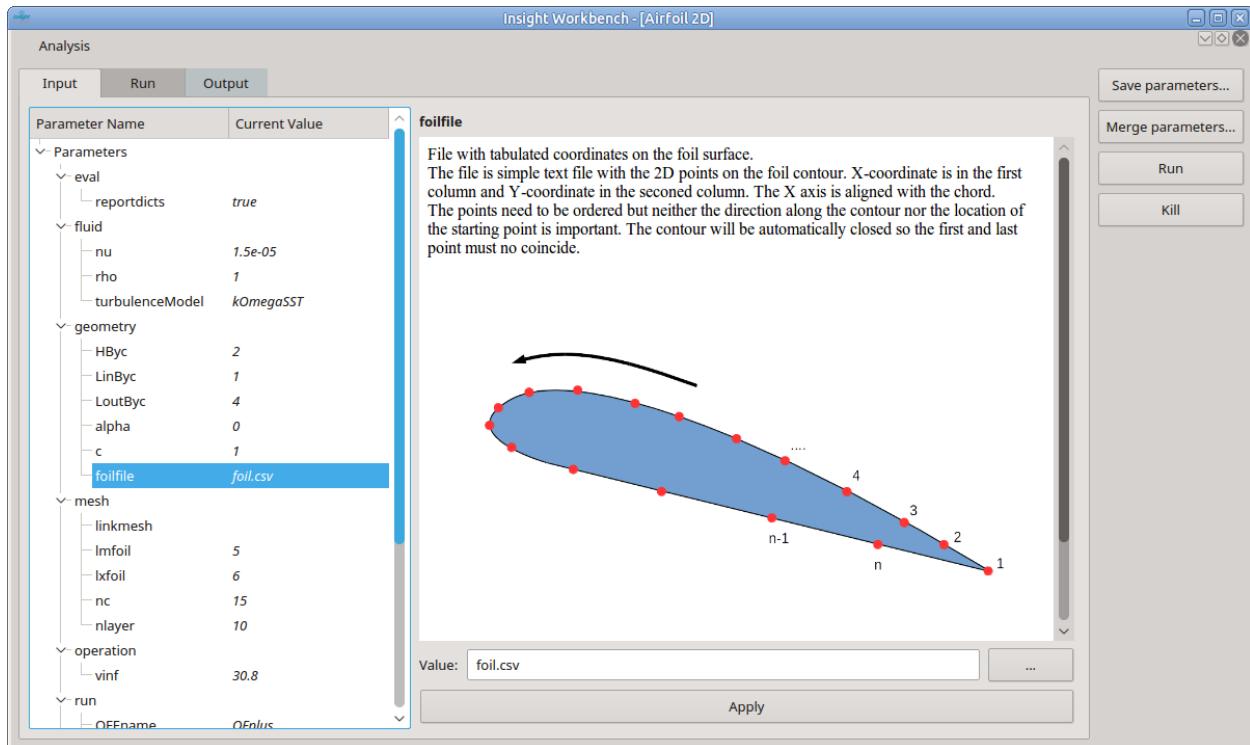


Figure 5: Parameter editing tab of the workbench

- the interruption appeared within a meshing procedure which includes intermediate meshes  
⇒ a mesh will be present but invalid
- the interruption appeared during the field initialization ⇒ a time directory is present but the fields are invalid,
- the interruption appeared during writing of a time directory ⇒ a time directory is present but invalid and the solver restart will fail.

If you are unsure about the validity of the case data, please consider to clean the case directory first and restart everything from the beginning.

**Launching Graphical Postprocessor ParaView** To inspect the results of a OpenFOAM simulation beyond the figure which the InsightCAE simulation extracts automatically from the simulation case, the independent graphical postprocessor Paraview can be used.

It can be launched from the command line using a script as explained in 6.4.1.

Alternatively, this can be done also from the workbench. On the right border of the analysis form, there is a button labelled "ParaView". For OpenFOAM-based simulation apps, this is enabled. Once it is pressed, Paraview will be launched in the selected working directory.

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

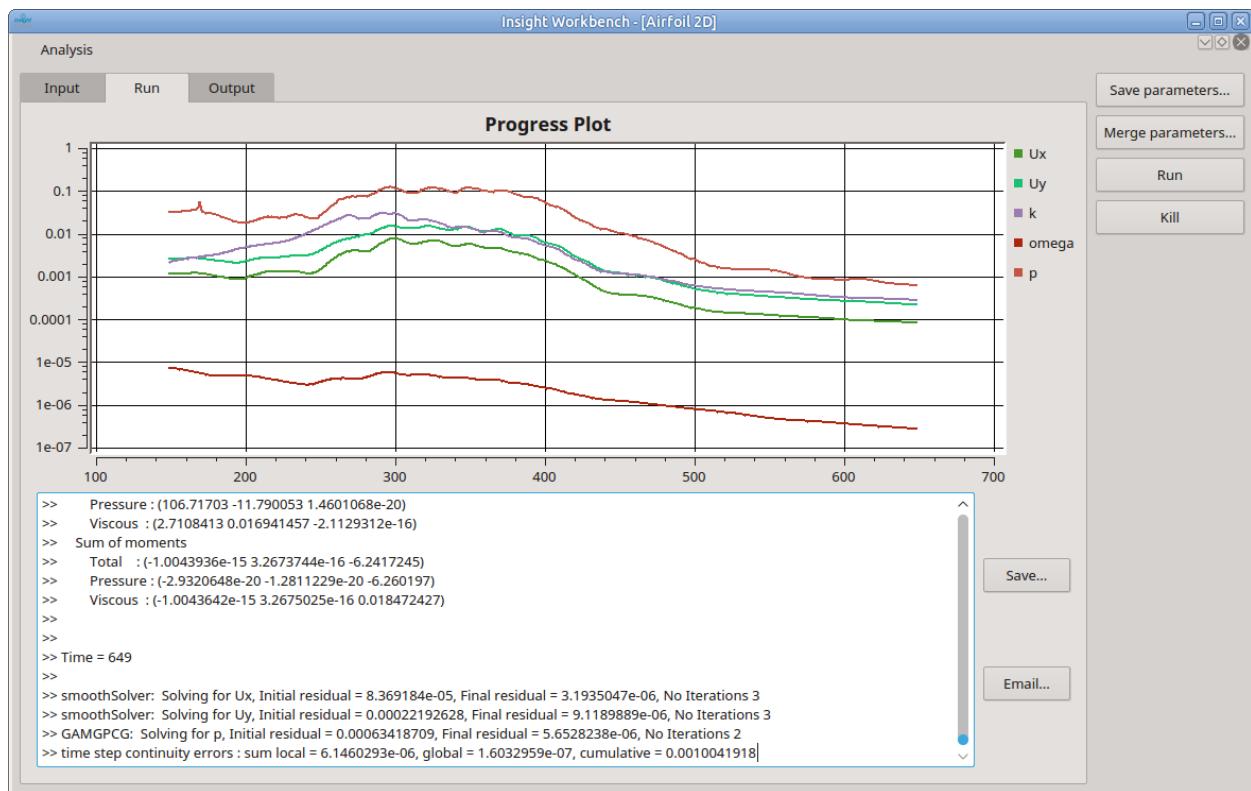


Figure 6: Solution progress tab of the workbench

### 5.2.3 Console on Headless Computers: analyze

## 5.3 Available Simulation Workflows

### 5.3.1 Numerical Wind Tunnel

### 5.3.2 Internal Pressure Loss

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

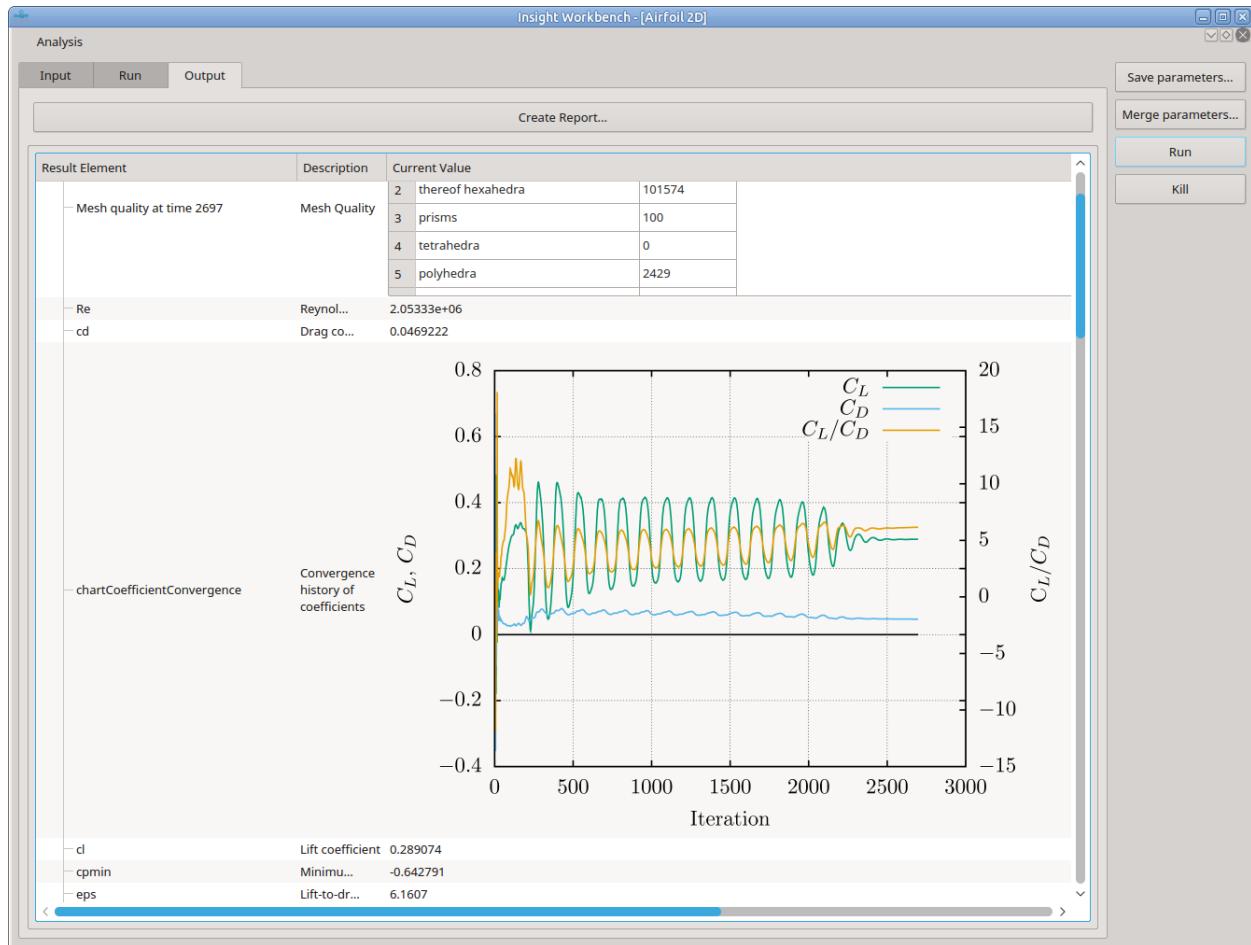


Figure 7: Result explorer tab of the workbench

## 6 Supporting Simulations with OpenFOAM

### 6.1 Case Setup GUI: isofCaseBuilder

### 6.2 Execution Monitor GUI: isofExecutionManager

### 6.3 Tabular Output Data Plot: isofPlotTabular

### 6.4 Common Tasks in the Context of OpenFOAM Simulations

#### 6.4.1 Controlling and Checking Running OpenFOAM Case Which Were Generated by InsightCAE

OpenFOAM cases which were generated by InsightCAE are enhanced with a number of additional features. These are utilized by the workbench frontend but can also be used to modify cases manually.

**Trigger Output Independently from the Configured Output Interval** The solver monitors the files in its case directory. If it finds a file named `wnow`, immediate output is triggered and this signal

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

file is removed by the solver.

This feature can be used to inspect the solution at any time while the solver is running using Paraview.

The file can e.g. created from the terminal:

```
1 $ touch wnow
```

Note: you might need to change to working directory to the case directory first.

**Trigger Immediate Output and Stop Solver Afterwards Without Raising an Error** The solver monitors the files in its case directory. If it finds a file named `wnowandstop`, immediate output is triggered, this signal file is removed by the solver and the solvers.

This feature can be used e.g. to gracefully end a solver run which is converged and the convergence is recognized by the user but not yet by the convergence monitoring feature of InsightCAE.

The file can e.g. created from the terminal:

```
1 $ touch wnowandstop
```

Note: you might need to change to working directory to the case directory first.

**Inspecting a Running Case Using Paraview** Some of the relevant numerical figures of a case are displayed in live updated charts in the GUI. But it is a very good habit to check the mesh and also the solution as soon as possible and well before valuable computing resource are wasted to compute inadequate solutions on bad meshes. If problems with the mesh or the solutions become obvious, it is possible to stop the analysis (e.g. by the "Kill" button in the workbench or by pressing CTRL+C in the terminal, where the analyze command line tool is running) and revise the settings.

The OpenFOAM cases can be loaded into Paraview as soon as they were created by one of the InsightCAE modules. It is perfectly possible to do this while the solver is running. It is also possible to trigger extra output if it is not yet available (see first paragraph of this section).

- When output<sup>5</sup> is there, Paraview is best launched by InsightCAE's wrapper script `isPV.py`:

```
1 $ isPV.py
```

Note: you might need to change to working directory to the case directory first.

- The wrapper automatically inserts the OpenFOAM case reader source into Paraview's visualization pipeline. You should consider the following settings:

1. Select at least all the mesh regions, which you would like to inspect. You can later filter out subsets of mesh regions. So it is ok, to select all.
2. Select the case type: either
  - "Reconstructed Case", if you performed a serial run or if the output has been already reconstructed from the processor directories (usually done for the last time step during the evaluation step)

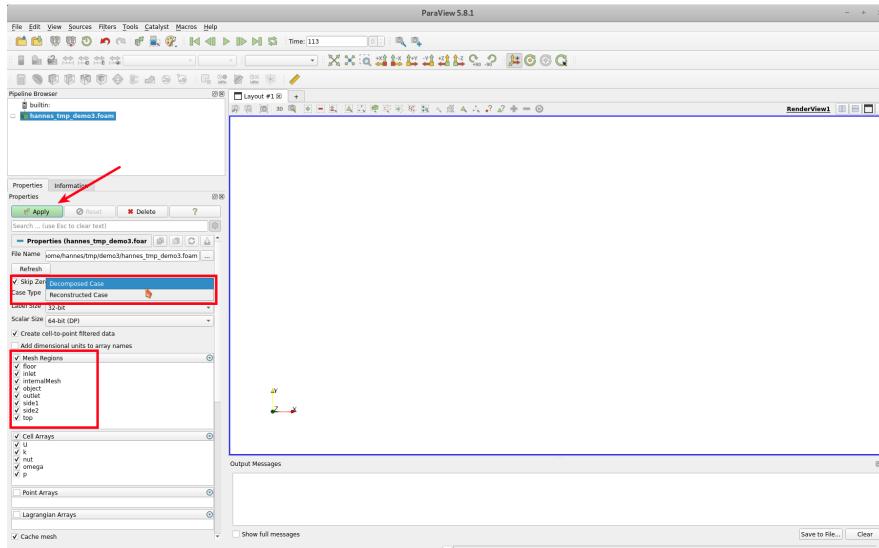
---

<sup>5</sup>Data of every output time instance is stored in a so-called time directory. These are directories in the case directory whose names are just numbers. The number correspond to the simulation time to which the data belongs. For a parallel run, these directories are created in the processor directories below the case directory and for a serial run, they are found directly in the top level case directory.

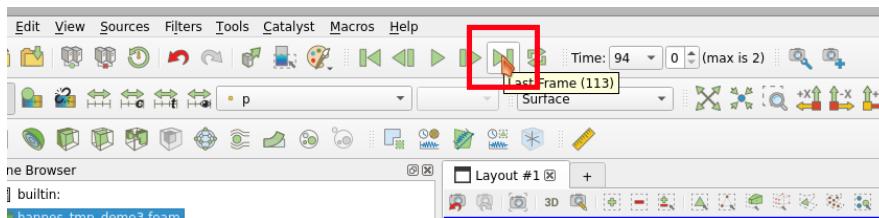
<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

- "Decomposed Case", if you want to monitor a parallel run.

When all settings are right, click on "Apply".



- Next, jump to the latest time step:

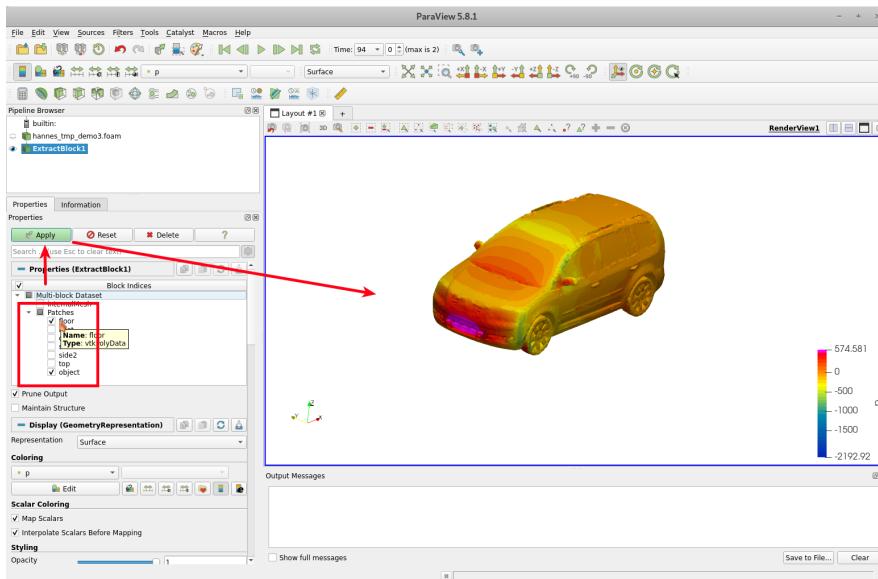


- Now, the scene can be set up using all the Paraview filters<sup>6</sup>.

Often, fields on boundary regions (e.g. walls) are of interest. These can be extracted using the "Extract Block" filter. To apply this filter, select from the menu **Filters** → **Alphabetical** → **Extract Block**. Make sure that the OpenFOAM case source is selected in the pipeline browser when the filter is added. When all the boundaries of interest are selected, click on "Apply".

<sup>6</sup>see <https://www.paraview.org/paraview-guide/> for Paraview's User Guide. It can be viewed online.

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------



#### 6.4.2 Meshing

**Extraction of Feature Edges from STL Geometries** Features edges can be extracted automatically from STLs. They are usually found by looking at the angle between neighbouring triangular facets. This method is usually automatically applied either in snappyHexMesh itself or within InsightCAE's workflows. This method works nicely for edges between planar surfaces. Though it fails e.g. for rounded edges like leading and trailing edges of propellers and foils. In such cases, it is required to extract feature edges manually and feed them into the meshing process.

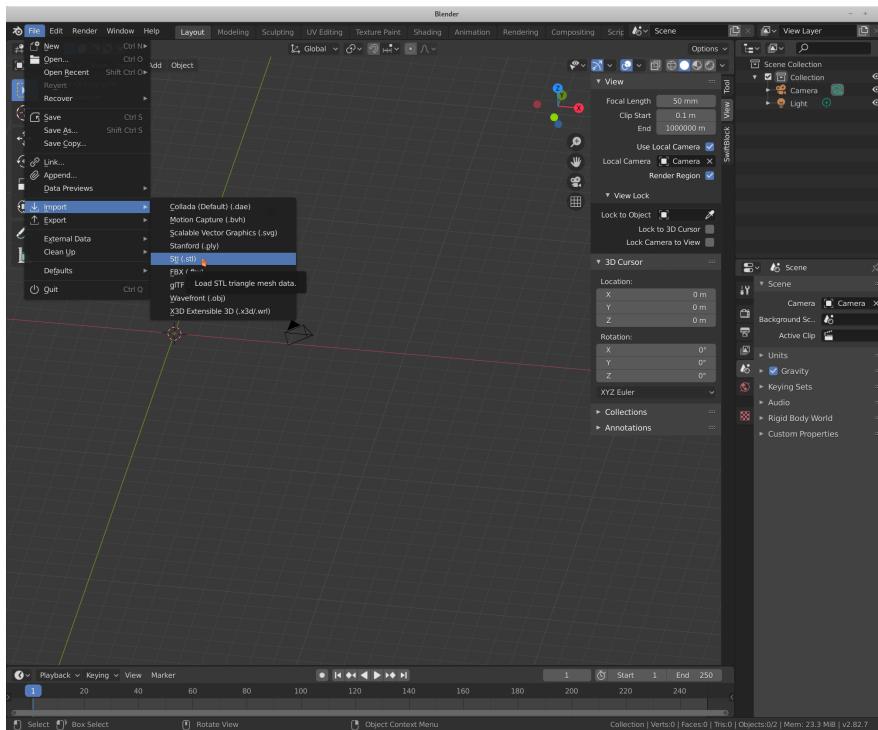
Here, it is described how to extract feature curves using the open source software Blender<sup>7</sup>. The following steps are required:

1. Import the geometry into Blender

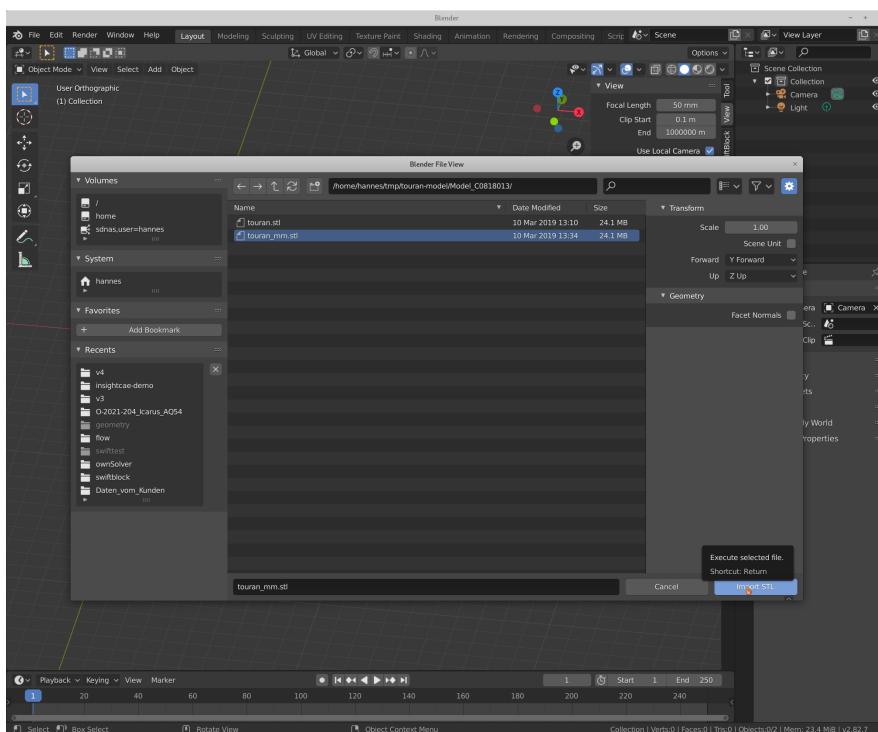
select in menu **File > Import > STL (\*.stl)**

<sup>7</sup><https://www.blender.org/>

Project Codeword	Document No	Editor	Date	Revision
doc	202002051012		November 24, 2021	0



## 2. select the STL file



## 3. make sure, the object is selected (orange line around the selected object is displayed)

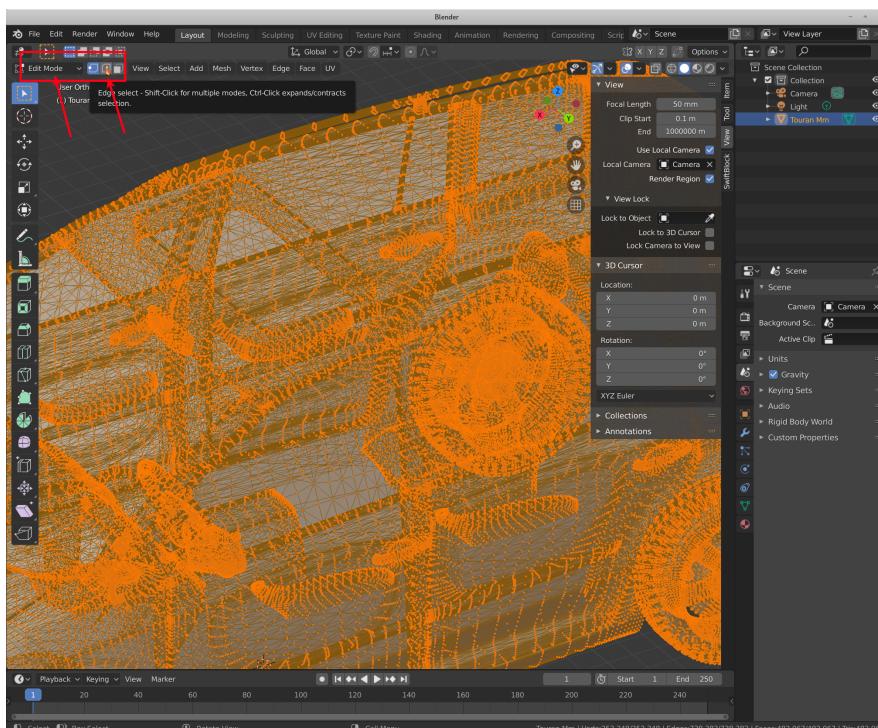
Objects can be selected by left click them with mouse.

Project Codeword	Document No	Editor	Date	Revision
doc	202002051012		November 24, 2021	0



4. switch to "Edit Mode", either by pressing the "Tab" key or by selecting the mode in the combo box in the upper left corner.

Next, switch to edge selection mode (second button right from the edit mode combo box)

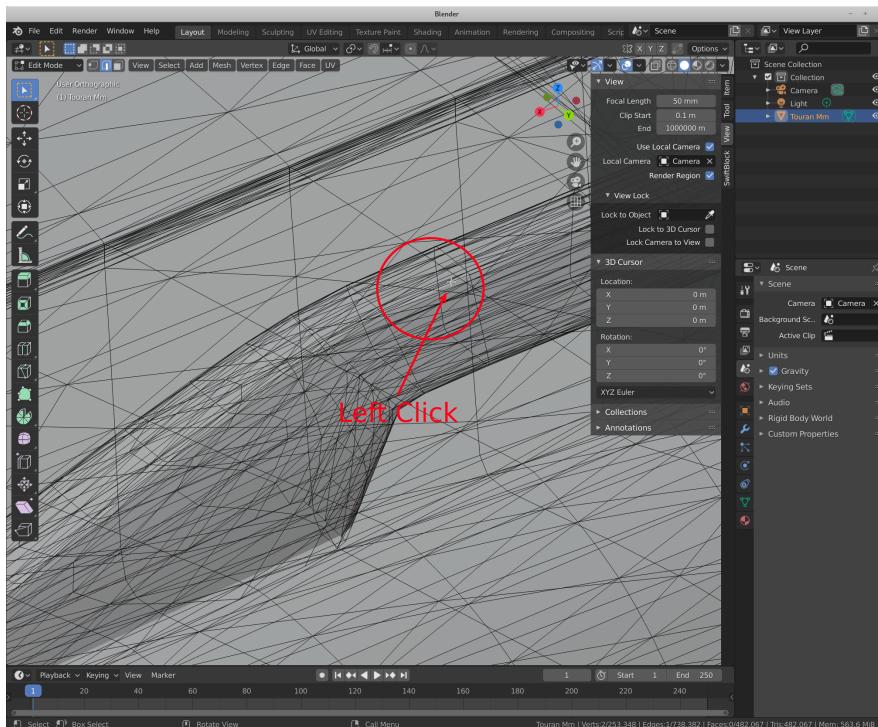


In this example, we want to select a line (chain of edges) along the left roof rail.

5. zoom into the location of the beginning of the line

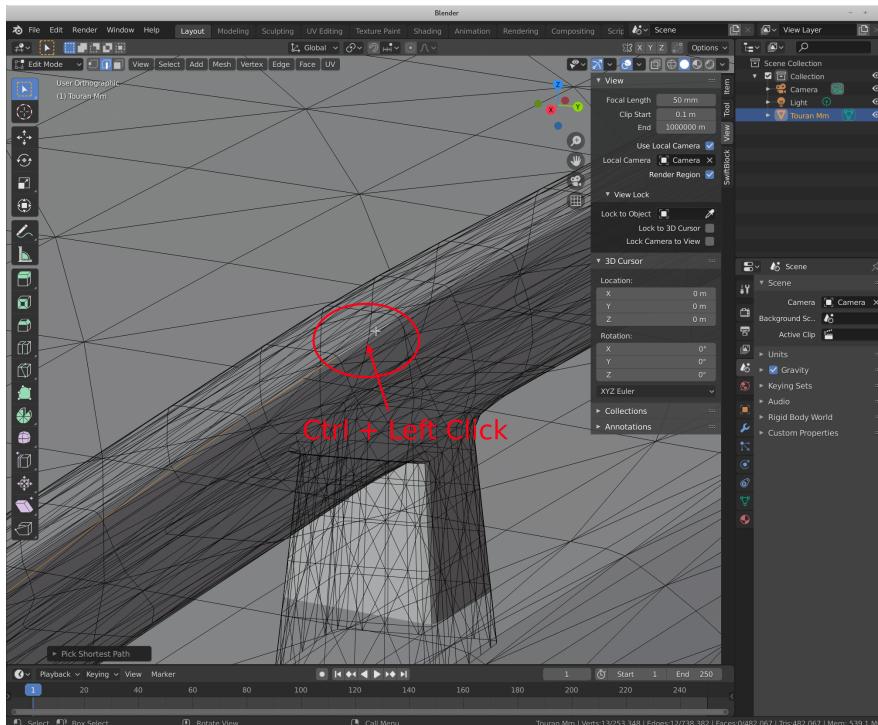
select the first edge with a left click on it.

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------



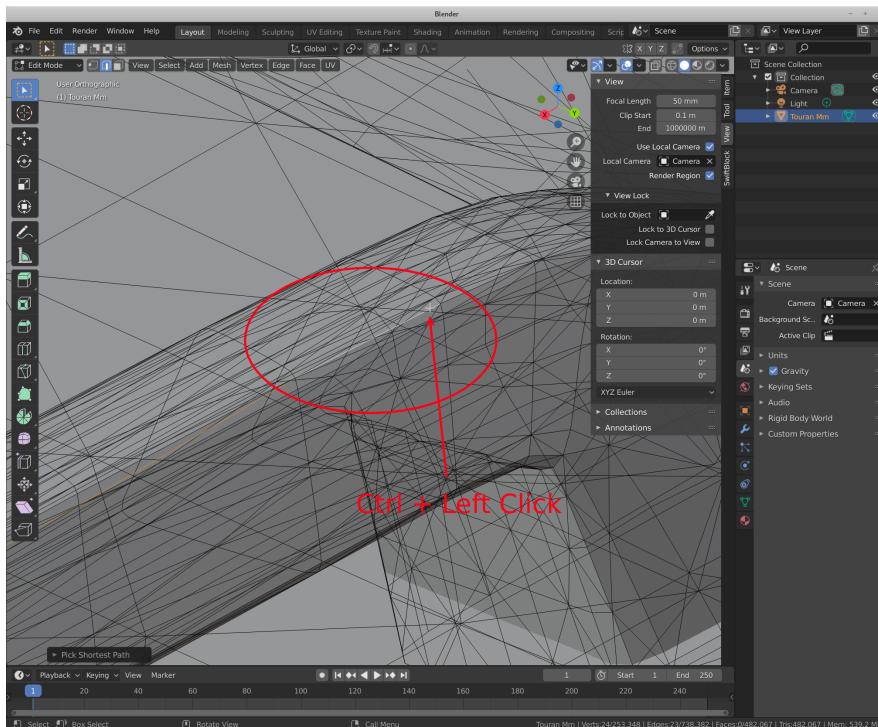
6. select another edge on the feature line: zoom to the next location and press Ctrl + Left click

The shortest path from the last selected edge and the current clicked edge will be added to the selection. To select edges as precise as possible, it might be required to click a number of intermediate locations. If a mistake is made, the last step can be undone by typing Ctrl+Z.



7. repeat the last step until the end of the feature line is reached.

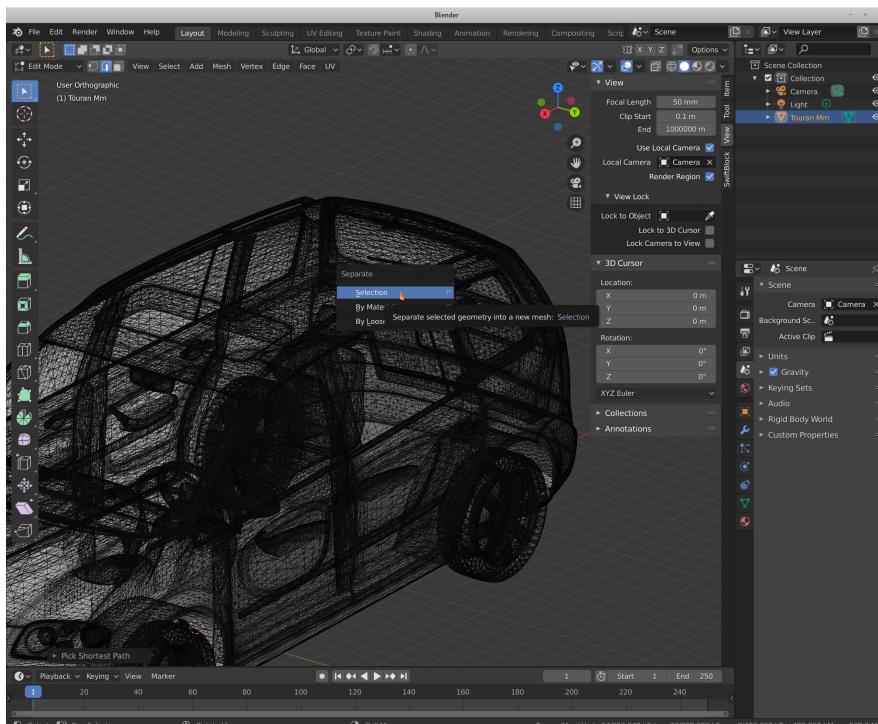
Project Codeword	Document No	Editor	Date	Revision
doc	202002051012		November 24, 2021	0



8. next create a new object from the selected edges by typing the key "P".

Then select "Selection" to create the new part from the current selection.

Once the selection is split off, another chain of edges can be selected by repeating the steps from 5.



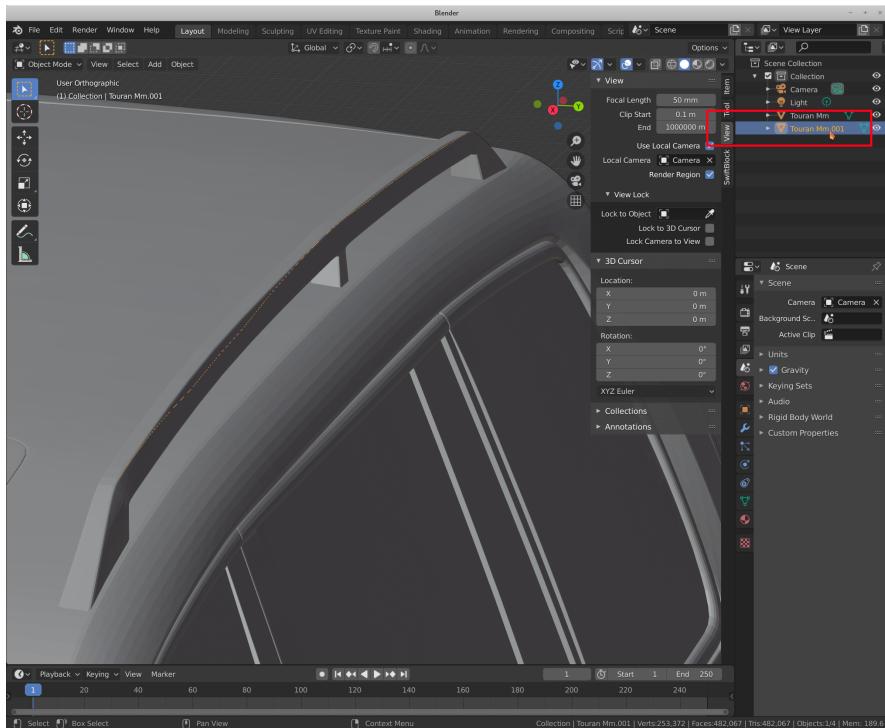
9. switch back to "Object Mode"

either by pressing the Tab key or by selecting "Object Mode" in the combo box in the upper left corner.

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

When multiple lines had been split into multiple new objects, these should be joined into a single one. To do so, select the objects in the object browser (upper right), then type "Ctrl+J".

Note: the mouse should be over the 3D viewport for Blender to accept the keyboard shortcut.



## 10. export the edges into an OBJ file.

Select the object in the object browser with a left click.

Then open the menu **File > Export > Wavefront (\*.obj)**.



<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

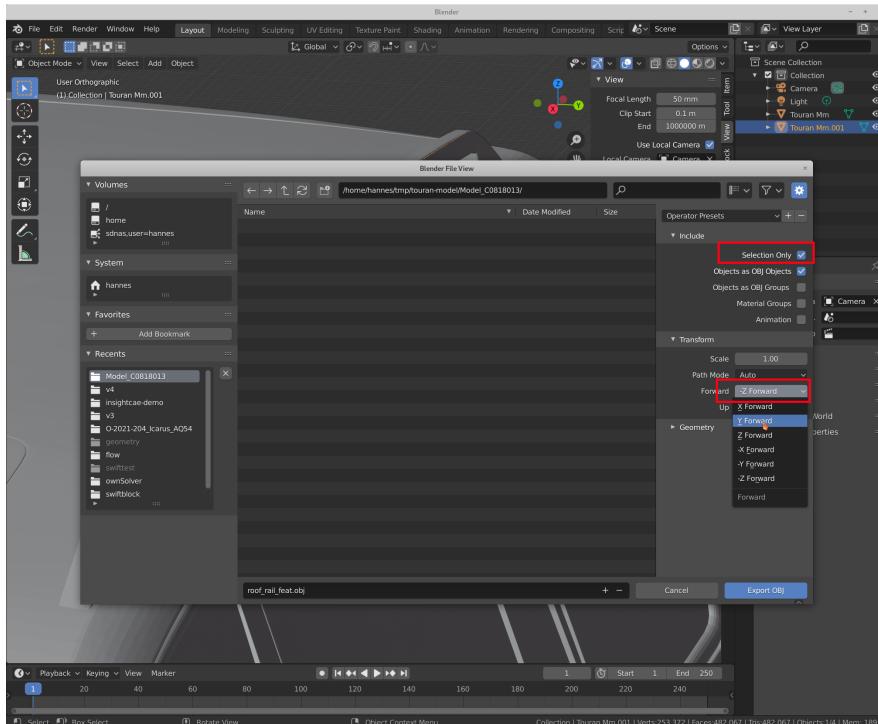
11. check the export settings. Make sure, the following is set:

- "Selection only" is checked

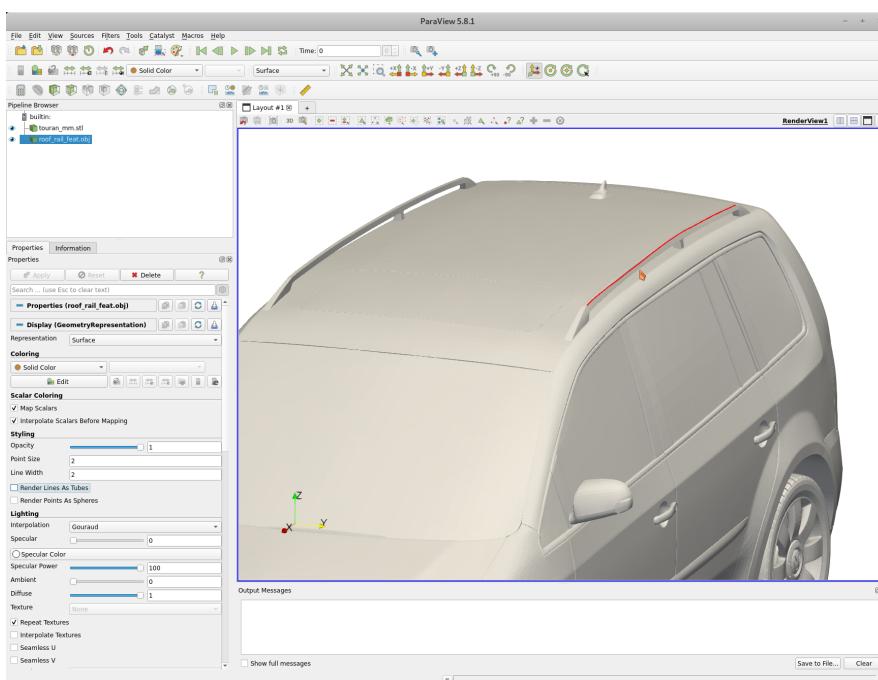
(Otherwise also the surface will be included in the exported file)

- In "Transform", "Forward" is set to "Y Forward"

(Otherwise the geometry will be wrongly oriented after export)



12. the resulting OBJ files can be loaded into Paraview and checked



13. Finally, the OBJ files can be converted into OpenFOAM's eMesh format. There is a tool called

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

"surfaceFeatureConvert" in the OpenFOAM toolbox, which can perform this task. The file type is recognized from the extension. If not already done, the OpenFOAM environment needs to be loaded (here using the alias of1806):

```
1 $ of1806  
2 $ surfaceFeatureConvert roof_rail_feat.obj roof_rail_feat.eMesh
```

## 7 Extensions to Back-End-Tools

### 7.1 OpenFOAM

## 8 InsightCAD

### 8.1 Introduction

InsightCAD is a script-based tool for creating three-dimensional geometry models. All geometric operations are based on the OpenCASCADE geometry kernel. It uses the boundary representation approach (BREP) for treating the geometry. Thus it is possible to import and export the common exchange formats IGES and STEP.

Although the primary intention of InsightCAD is creation of fully parameterized CAD models for systematic numerical simulations, it can also be used for mechanical design purposes. It therefore provides the possibility to export projections and sections of the created models in DXF format for use in drawings. Additionally, there is support for a library of parametric standard parts.

### 8.2 Basic Concept

The basic entity in InsightCAD is a "model". A model is described by a script and stored in an ASCII script file (extension ".iscad"). Inside a model script, symbols are defined, which can represent the following data types:

- Scalars
- Vectors
- Datum objects (axes, planes)
- 3D geometry objects (features)
- Selections of vertices, edges, faces or solids of 3D geometry objects

There is no explicit type declaration: the data type of each symbol is deduced from the defining expression.

Beyond their geometry representation, geometry feature objects are containers for scalar, vector, datum and feature objects. These symbols can be accessed in the model script but are read-only.

In a model script, after the definition of the aforementioned symbols, an optional section with postprocessing actions can follow. These can be e.g. file export, drawing export or others.

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

**General CAD Model Script Syntax** The general model script layout begins with a mandatory symbol defining section, followed by an optional postprocessing section, started by "@post":

```

1 <identifier> [ = | ?= | : ] <expression>;
2 ...
3
4 @post
5
6 <postprocessing action>;
7 ...

```

Comments are lines starting with "#" or text regions enclosed by "/\*" and "\*/" (C-style).

## 8.3 Submodels and Subassemblies

It is possible to load another model into the current one by the "loadmodel" command (see section "Features"). In this case, the loaded model represents a subassembly, i.e. a compound of features. Since not all defined geometry objects in the submodel may represent assembly components, marking of components is supported by the feature definition syntax and needs to be utilized properly in the definition script of the submodel. Also, when loading models (subassemblies), parameters can be passed to the submodel. These can be scalars, vectors, datums or features.

## 8.4 Symbol Definition

### 8.4.1 Scalars

Example:

```

1 D ?= 123.5;
2 L = 2.*D;

```

The "?=" operator assigns a default value. This needs to be used for symbols which are intended to be used as parameters in the "loadmodel" feature command. Symbols defined by the equal sign operator ("=") cannot be overridden during the loadmodel command.

Supported operations are listed in table 3.

There are scalars predefined in each model. They are included in table 6.

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

InsightCAD script	Description
+ - * /	basic algebra
mag(<vector>)	magnitude of vector
sqrt(<scalar>)	square root
sin(<scalar>)	sin(x)
cos(<scalar>)	cos(x)
tan(<scalar>)	tan(x)
asin(<scalar>)	arcsin(x)
acos(<scalar>)	arccos(x)
ceil(<scalar>)	smallest following integer
floor(<scalar>)	largest previous integer
round(<scalar>)	round to next integer
pow(<scalar:a>, <scalar:n>)	a^n
atan2(<scalar:y>, <scalar:x>)	arctan(y/x)
atan(<scalar>)	arctan(x)
volume(<feature>)	volume of feature
cumedgelen(<feature>)	sum of all edges lengths
<vector>.x	x component of vector
<vector>.y	y component of vector
<vector>.z	z component of vector
<feature> \$ <identifier>	scalar property of feature
<vector> & <vector>	Scalar product

Table 3: Algebraic operations in ISCAD scripts

#### 8.4.2 Vectors

Example:

```
1 v ?= 5*EX + 3*EY + [0,0,1];
2 ev = v/mag(v);
```

The "?=" operator assigns a default value. This needs to be used for symbols which are intended to be used as parameters in the "loadmodel" feature command. Symbols defined by the equal sign operator ("=") cannot be overridden during the loadmodel command.

Supported operations are listed in table 4.

There are vectors predefined in each model. They are included in table 6.

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

InsightCAD script	Description
+ -	basic algebra
[⟨x⟩, ⟨y⟩, ⟨z⟩]	vector from components
⟨feature⟩@⟨vector⟩	vector property of feature
⟨scalar⟩*⟨vector⟩	scaled vector
⟨vector⟩/⟨scalar⟩	scaled vector
⟨vector:a⟩^⟨vector:b⟩	Cross product $\vec{a} \times \vec{b}$
bbmin(⟨feature⟩)	minimum corner of feature bounding box
bbmax(⟨feature⟩)	maximum corner of feature bounding box
cog(⟨feature⟩)	center of gravity coordinates of feature
refpt(⟨datum⟩)	reference point of datum (base point of axis or plane)
refdir(⟨datum⟩)	reference direction of datum (direction of axis or normal of plane)

Table 4: Vector operations and functions in ISCAD scripts

#### 8.4.3 Datums

Some simple examples are given below.

```

1 myaxis ?= RefAxis(0, EX+EY); # diagonal axis
2 myplane = Plane(5*EX, EY); # offset plane
3 myplane2 = XZ << 5*EX; # same offset plane
4 axis2 = xsec_plpl(XY, myplane); # axis at intersection of XY-Plane and offset
      plane

```

The "?=" operator assigns a default value. This needs to be used for symbols which are intended to be used as parameters in the "loadmodel" feature command. Symbols defined by the equal sign operator ("=") cannot be overridden during the loadmodel command.

Supported operations are listed in table 5.

InsightCAD script	Description
⟨feature⟩%⟨identifier⟩	Access datum inside another feature.
⟨datum⟩ « ⟨vector:Δ⟩	Copy of datum, translated by $\vec{\Delta}$
Plane(⟨vector:p0⟩, ⟨vector:n⟩)	Datum plane with origin p0 and normal n.
SPlane(⟨vector:p0⟩, ⟨vector:n⟩, ⟨vector:e_up⟩)	Additionally, the y-direction of the plane CS is aligned with $\vec{e}_{up}$ .
RefAxis(⟨vector:p0⟩, ⟨vector:e_x⟩)	Axis with origin p0 and direction $\vec{e}_x$ .
xsec_axpl(⟨datum:ax⟩, ⟨datum:pl⟩)	Datum point at intersection between axis ax and plane pl
xsec_plpl(⟨datum:pl1⟩, ⟨datum:pl2⟩)	Datum axis at intersection between plane pl1 and pl2
xsec_ppp(⟨datum:pl1⟩, ⟨datum:pl2⟩, ⟨datum:pl3⟩)	Datum point at intersection between three planes

Table 5: Vector operations and functions in ISCAD scripts

There are datums predefined in each model. They are listed in table 6.

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

InsightCAD script	Description
M_PI	$\pi$
deg	Conversion factor from degrees to radians ( $180/\pi$ )
EX	Unit vector in X direction $\vec{e}_x = (1 \ 0 \ 0)^T$
EY	Unit vector in Y direction $\vec{e}_y = (0 \ 1 \ 0)^T$
EZ	Unit vector in Z direction $\vec{e}_z = (0 \ 0 \ 1)^T$
O	Origin $\vec{O} = (0 \ 0 \ 0)^T$
XY	X-Y-Plane
XZ	X-Z-Plane
YZ	Y-Z-Plane

Table 6: Predefined symbols (scalars, vectors and datums) in ISCAD scripts

#### 8.4.4 Features

A very simple example is given below. It consists of two primitive features (cylinder) and a boolean operation (subtraction).

```

1 tool = Cylinder(-10*EY, 10*EY, 2);
2
3 pierced_cylinder:
4 Cylinder(0, 20*EX, 10, centered)
5 -
6 tool;
```

Geometry symbols can be defined by a "=" or a ":" operator. The difference comes from a possible use of the model as a subassembly later on. Since usually not all defined features in a model are assembly components but some are only intermediate modeling steps, there are these two syntaxes for defining a feature with a subtle difference: "<identifier> = <expression>;" defines an intermediate feature while "<identifier>: <expression>;" does the same geometry operation but marks the result as being an assembly component. In the above example, only the feature "pierced\_cylinder" is marked as a component. Thus if the above example would be loaded as a subassembly, only the "pierced\_cylinder" will be shown and included in e.g. mass calculations.

InsightCAD script	Description
<feature:a> - <feature:b>	Boolean subtract of feature b from feature a
<feature:a>   <feature:b>	Boolean unite of feature a and feature b
<feature:a> & <feature:b>	Boolean intersection of feature a and feature b
<feature> « <vector:delta>	Copy of feature, translated by vector delta
<feature> * <scalar:s>	Copy of feature, scaled by s (relative to global origin O)
<feature>.<identifier:subfeatname>	Access of subfeature

Table 7: Vector operations and functions in ISCAD scripts

#### 8.5 Feature Commands

In this section, an incomplete subset of the available feature commands is described in detail. For a complete list, please refer to the online documentation in the iscad editor (press Ctrl+F).

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

### 8.5.1 Transformation

Transform(<feature:f>, <vector:delta>, <vector:phi>)

Transformation of feature f. Translation by vector delta and rotation around axis vector phi (magnitude of phi gives rotation angle).

Place(<feature:f>, <vector:p\_0>, <vector:e\_x>, <vector:e\_z>)

Places the feature f in a new coordinate system. The new origin is at point  $\vec{p}_0$ , the new x-axis along vector  $\vec{e}_x$  and the new z-direction is  $\vec{e}_z$ .

### 8.5.2 Import

import(<path>)

Imports solid geometry from a file. The format is recognized from the filename extension. Supported formats are IGS, STP, BREP.

Sketch(<datum:pl>, <path:file>, <string:name> [, <identifier>=<scalar>, ... ])

Reads a sketch (i.e. a singly closed contour) from a file. The geometry in the sketch is expected to be drawn in the X-Y-Plane. It is placed on the given plane pl. Sketch file format is recognized from the file name extension. Supported are ".dxf" and ".fcstd" (FreeCAD). The name is interpreted as layer name in DXF and sketch name in FreeCAD files.

For FreeCAD sketches, a list of parameter values can optionally be supplied. Upon loading, the sketch will be regenerated through FreeCAD with these values.

loadmodel( <identifier:modelname> [, <identifier> = <feature>|<datum>| <vector>|<scalar>, ... ] )

Parses another InsightCAD model (submodel) and inserts a compound of all features into the current model, which were marked as components in the submodel (i.e. which were defined with the colon ":" operator instead of the equal sign "=" in the submodel).

The model filename has to be "<modelname>.iscad". It is searched for in the following directories:

1. the directories listed in the environment variable "ISCAD\_MODEL\_PATH" (separated by ":")
2. the subdirectory "iscad-library" in InsightCAEs shared file directory
3. in the current directory

Optionally, a list of symbols is inserted into the namespace of the submodel (additional optional parameters).

### 8.5.3 Geometry Construction

**Primitives** There are commands for creation of several different geometrical primitives, e.g.

- 1D: Arc, Line, SplineCurve
- 2D: Quad, Tri (triangle), RegPoly (regular polygon), Circle, SplineSurface
- 3D: Sphere, Cylinder, Box, Bar, Cone, Pyramid, Torus

Extrusion(<feature:f>, <vector:L> [, centered ] )

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

Extrude the feature f with direction and length vector L. When the keyword "centered" is given, the extrusion is centered around f.

```
Revolution( <feature:f>, <vector:p_0>, <vector:axis>, <scalar:phi> [, centered] )
```

Creates a revolution of the planar feature f. The rotation axis is specified by origin point  $\vec{p}_0$  and the direction vector axis. Revolution angle is specified by phi. By giving the keyword "centered", the revolution is created symmetrically around the base feature.

**Other Feature Commands** There are more features available. A comprehensive list is obtained by pressing Ctrl+F in the iscad editor.

## 8.6 Lower Dimensional Shape Selection

ISCAD supports rule based selection of lower dimensional features (i.e. edges or faces of a solid). The selection is generated by a selection command: a question mark, followed by the type of shape to query. The result is a selection object:

```
1 <feature expression|feature selection>?(vertices|edges|faces|solids);('<command_
  string>' [, parameter 0 [, ..., parameter n] ] )
```

It is possible to supply additional arguments to the selection expression, like scalars, vectors or features.

An example: the following expression selects the circumferential face of the cylinder c (all faces, which are not plane) and stores the selection in "shell\_faces":

```
1 c = Cylinder(0, 5*EZ);
2 shell_face = c ? faces('!isPlane');
3 min_end_face = c ? faces('isPlane&&minimal(CoG.z)');
```

The selection command string contains rules for the selection. Finally, the command string is evaluated as a boolean expression comprising comparison operators, boolean operators and query functions. Within these boolean expressions, quantity functions can be used. The available set of query functions and quantity functions depends on the type of shape which shall be queried.

Boolean expressions available for all kinds of lower dimensional shapes are listed in table 8.

Quantity functions available for all kinds of lower dimensional shapes are listed in table 9.

Command	Description
<code>==, &lt;, &gt;, &gt;=, &lt;=</code>	value comparison
<code>!</code>	not
<code>&amp;&amp;</code>	and
<code>  </code>	or
<code>&lt;value 1&gt; ~ &lt;value 2&gt; { &lt;tolerance&gt; }</code>	approximate equality
<code>in(&lt;selection set&gt;)</code>	true if shape is in other selection
<code>maximal(&lt;quantity&gt;)</code>	true for the shape with maximum quantity
<code>minimal(&lt;quantity&gt;)</code>	true for the shape with minimum quantity

Table 8: General boolean functions and operators available for all kinds of lower dimensional shape

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

Command	Description
angleMag(<vec 1>, <vec 2>)	angle between vec 1 and vec 2
angle(<vec 1>, <vec 2>)	angle between vec 1 and vec 2
%d<index>	parameter <index> as scalar
%m<index>	parameter <index> as vector
%<index>	parameter <index> as selection set

Table 9: General quantity functions available for all kinds of lower dimensional shape

### 8.6.1 Vertices

There are no special boolean functions or operators for vertices.

The available quantity functions for vertices are listed in table 10.

Command	Description
loc	location of the vertex

Table 10: Quantity functions available for vertex selections

### 8.6.2 Edges

Boolean functions for edges are listed in table 11.

The available quantity functions for edges are listed in table 12.

Command	Description
isLine	true, if edge is straight
isCircle	true, if edge is circular
isEllipse	true, if edge is elliptical
isHyperbola	true, if edge is on a hyperbola
isParabola	true, if edge is on a parabola
isBezierCurve	true, if edge is a bezier curve
isBSplineCurve	true, if edge is a BSpline curve
isOtherCurve	true, if edge is none of the above
isFaceBoundary	true, if edge is boundary of some face
boundaryOfFace(<set>)	true, if edge is boundary of one of the faces in set
isPartOfSolid(<set>)	true, if edge is part of one of the solids in set
isCoincident(<set>)	true, if edge is coincident with one of the edges in set
isIdentical(<set>)	true, if edge is identical with one of the edges in set
projectionIsCoincident(<set>, <vec:p0>, <vec:n>, <vec:up>, <scalar:tol>)	true, if projection of edge is coincident with some edge in set

Table 11: Boolean functions available for edge selections

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

Command	Description
len	length of edge
radialLen(<vec:ax>, <vec:p0>)	radial distance between ends with respect to axis (p0,ax)
CoG	center of gravity of edge
start	start point coordinates
end	end point coordinates

Table 12: Quantity functions available for edge selections

### 8.6.3 Faces

Boolean functions for faces are listed in table 13.

The available quantity functions for faces are listed in table 14.

Command	Description
isPlane	true, if is a plane
isCylinder	true, if is a cylindrical surface
isCone	true, if is a conical surface
isSphere	true, if is a spherical surface
isTorus	true, if is a toroidal surface
isBezierSurface	true, if is a bezier surface
isBSplineSurface	true, if is a BSpline surface
isSurfaceOfRevolution	true, if is a surface of revolution
isSurfaceOfExtrusion	true, if is a surface of extrusion
isOffsetSurface	true, if is a offset surface
isOtherSurface	true, if is some other kind of surface
isPartOfSolid(<set>)	
isCoincident(<set>)	
isIdentical(<set>)	
adjacentToEdges(<set>)	
adjacentToFaces(<set>)	

Table 13: General boolean functions available for face selections

Command	Description
area	area of the face
CoG	center of gravity of the face
cylRadius	radius of a cylindrical face
cylAxis	axis direction of a cylindrical face

Table 14: Quantity functions available for face selections

### 8.6.4 Solids

There are no special boolean functions or operators for solids.

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

The available quantity functions for solids are listed in table 15.

Command	Description
CoG	center of gravity
volume	volume of the solid

Table 15: Quantity functions available for solid selections

## 8.7 Postprocessing Actions

### 8.7.1 Drawing Export

```
DXF(<path:outputfile>) « <feature:f> <view_definition> [, <view_definition>, ... ]
```

The DXF postprocessing action creates a DXF file for further use in drawings. Several views are derived from the feature f. A <view\_definition> takes the following form:

```
1<identifier:viewname> (
2  <vector:p_0>, <vector:n>, up <vector:e_up>
3  [, section]
4  [, poly]
5  [, skiphl]
6  [, add [l] [r] [t] [b] [k] ]
7 )
```

It defines a view on the point vector  $\vec{p}_0$  with normal direction vector  $\vec{n}$  of the view plane. The upward direction (Y-direction) is aligned with vector  $\vec{e}_{up}$ .

The keyword "section" toggles whether only the outline is projected or if the view plane creates a section through the geometry.

If keyword "poly" is given, the DXF geometry will be discretized. This is more robust but creates much larger DXF files.

Keyword "skiphl" toggles whether hidden lines are output.

The keyword "add" followed by the key letters l, r, t, b and/or k enables creation of additional projections from the left, right, top, bottom and/or back, respectively.

An example is given below:

```
1c: Cylinder(0, 100*EX, 20);
2
3@post
4
5DXF("c.dxf") << c
6    top ( 0, EX, up EY )
7    front ( 0, EZ, up EY )
8;
```

### 8.7.2 Mesh Creation

```
gmsh(<path:outputfile>) « <feature:f> as <identifier:l> <mesh_parameters>
```

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

Generates a (triangular or tetrahedral) mesh for an FEA analysis of the feature f. Gmsh is used as a meshing backend. The mesh format is determined by the outputfile extension (.med = MED format). The label of the mesh is set to l.

The syntax of the <mesh\_parameters> are as follows:

```

1 L = ( <scalar:Lmax> <scalar:Lmin> )
2 [linear]
3 vertexGroups( [ <identifier:group name> = <vertex set> [ @ <scalar:size> ], ... ]
4 )
5 edgeGroups( [ <identifier:group name> = <edge set> [ @ <scalar:size> ], ... ] )
6 faceGroups( [ <identifier:group name> = <face set> [ @ <scalar:size> ], ... ] )
7 [ vertices ( [ <identifier:vertex name> = <vector:location> ], ... ) ]

```

The general mesh size is set by Lmax and Lmin. The optional keyword "linear" switch from quadratic to linear elements. Named groups of vertices, edges and faces can be created using the keywords "vertexGroups", "edgeGroups" and "faceGroups" repectively. Each group definition takes the form "groupname" = "selection set" (see section "Lower dimensional shape selection" for definition of selection sets). Optionally, a mesh size can be assigned to each defined group by appending an @ sign followed by a scalar value.

An example is given below:

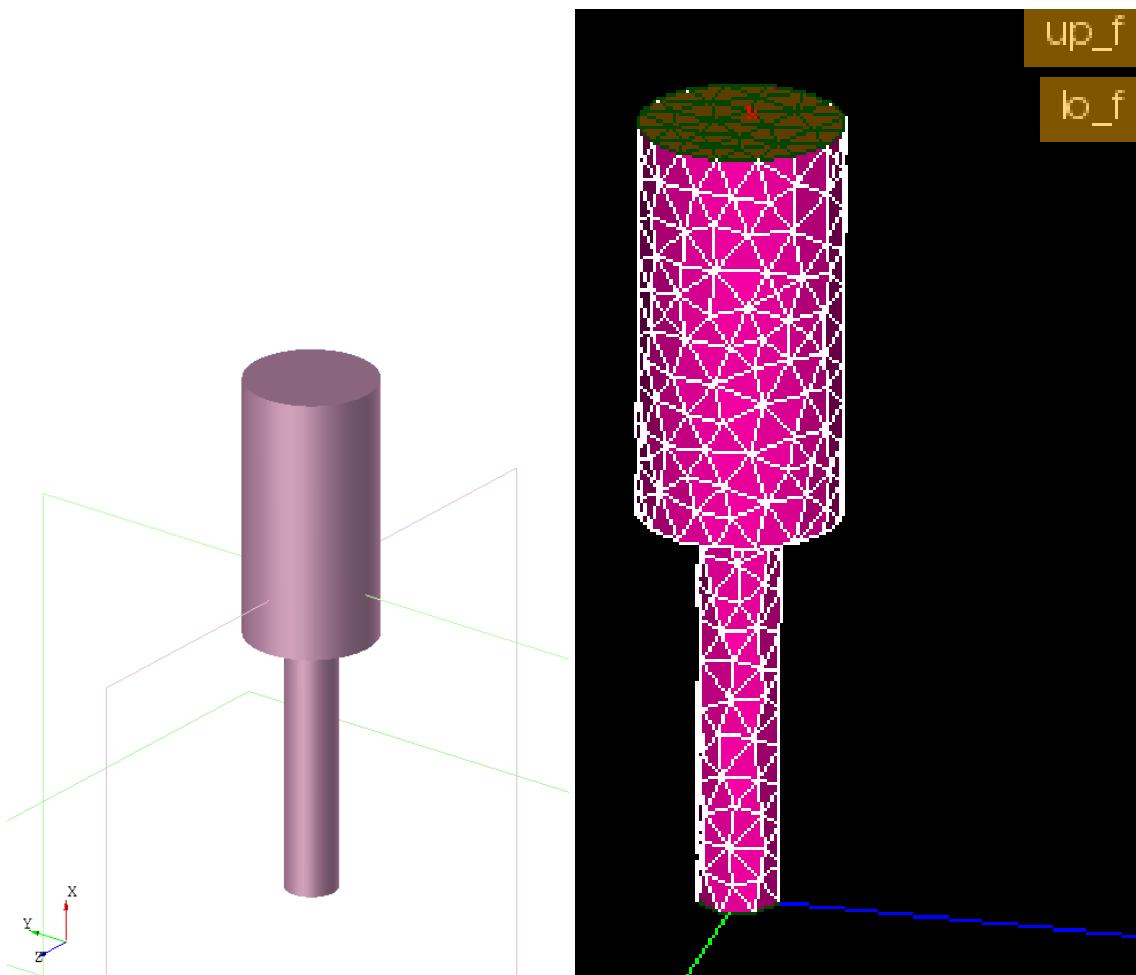
```

1 c:
2 Cylinder(0, 100*EX, 20)
3 |
4 Cylinder(100*EX, ax 100*EX, 50)
5 ;
6
7 @post
8
9 gmsh("c.med") << c as cyl
10     L = (10 0.1)
11     linear
12     vertexGroups()
13     edgeGroups()
14     faceGroups(
15         lo_f = c?faces('isPlane&&minimal(CoG.x)')
16         up_f = c?faces('isPlane&&maximal(CoG.x)')
17     )
18 ;

```

Result: ISCAD model (left) and resulting mesh (right):

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------



## 8.8 Graphical Editor ISCAD

ISCAD is a graphical editor for InsightCAD scripts. It consists of a text editor for editing the script contents and a 3D view and some other elements for inspecting the resulting model. A screenshot is shown in the figure below.

The model script is entered into the text editor widget right of the 3D display. Once a script shall be evaluated, it can be parsed by clicking in the button "Rebuild" or pressing Ctrl+Return.

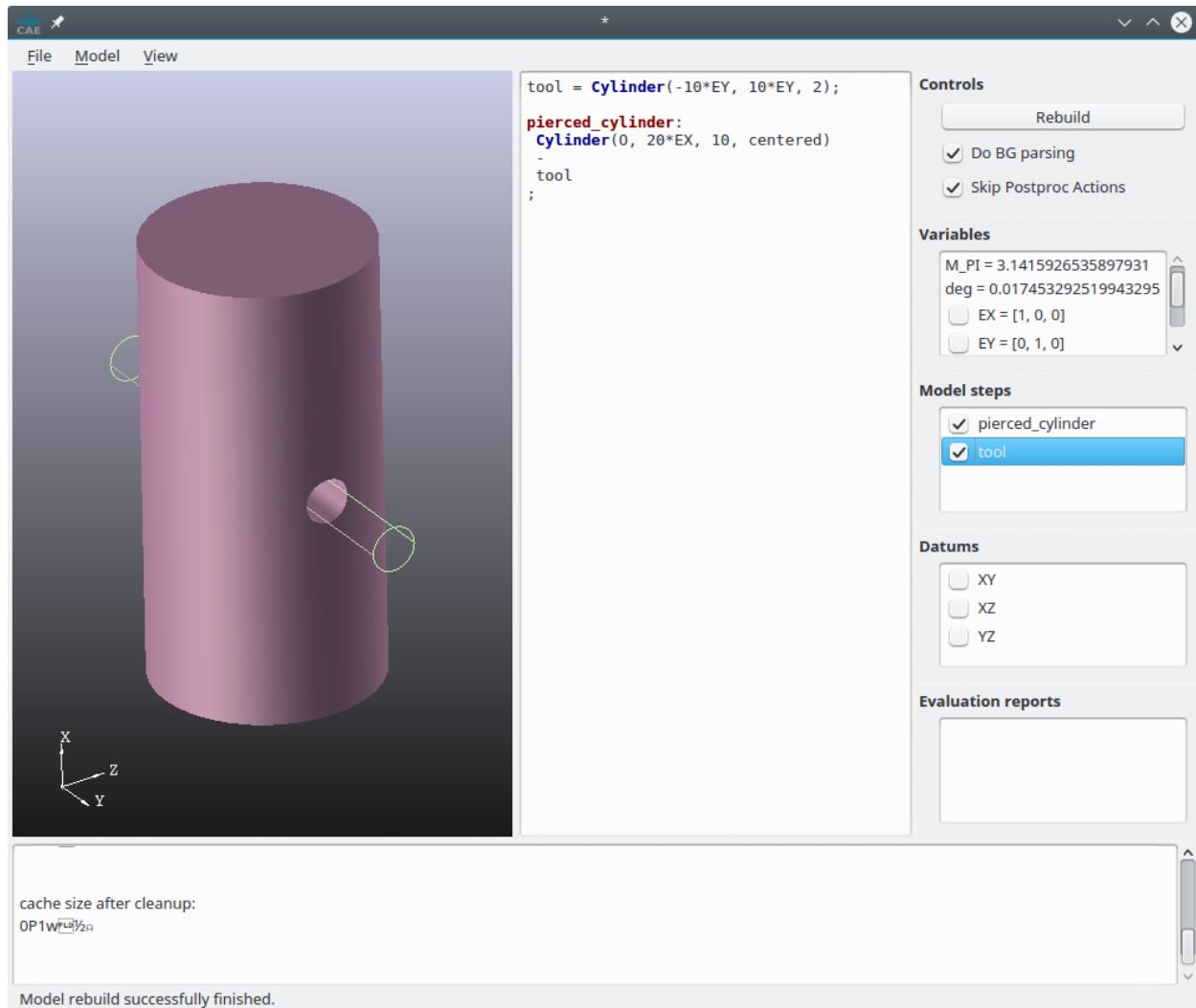
After parsing the model, the following results are displayed:

- A list of all created feature symbols in the "Model Steps" list box. If the check box is checked, the 3D geometry is displayed in the 3D view.
- The values of all scalars and vectors in the "Variables" list box.

For each vector variable, a check box is displayed. If it is checked, the vector is interpreted as a point location and the point is shown in the 3D display window.

- All defined datums are listed in the "Datums" list box. Again, the check box controls, whether the datum is displayed in the 3D view.

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------



### 8.8.1 3D Graphics Display

#### View Manipulation

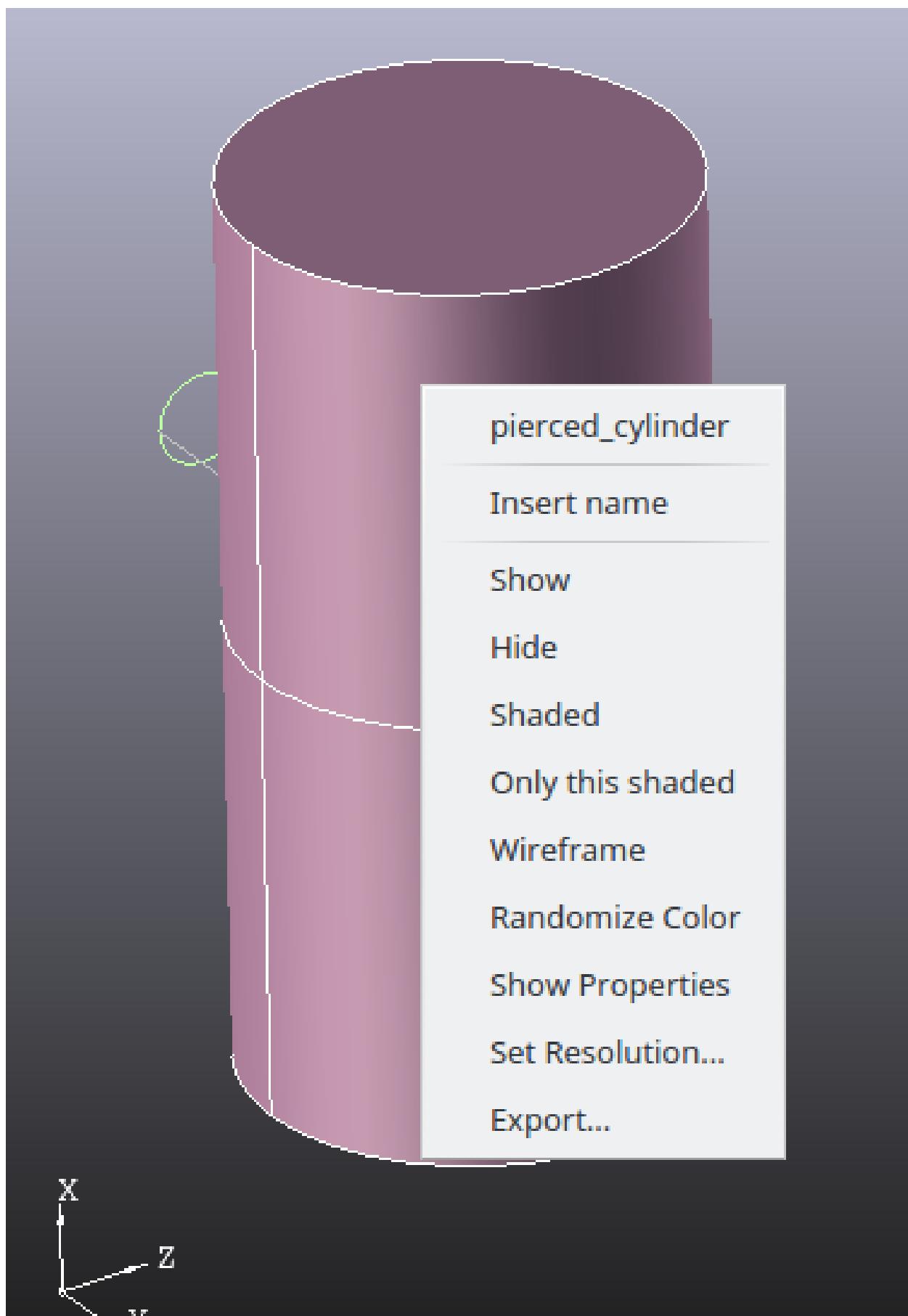
- Dragging: Shift + mouse move
- Scaling: Ctrl + horizontal mouse move
- Rotating: Alt + mouse move

**Model Navigation** When hovering the mouse pointer over a displayed feature in the 3D geometry window, it is highlighted. The highlighted feature is the one, which would be selected during subsequent mouse clicks.

When the left mouse button is pressed, the highlighted feature is selected and all its contained reference points are displayed.

When the right mouse button is pressed, a context menu for the selected feature is displayed.

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------



<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------

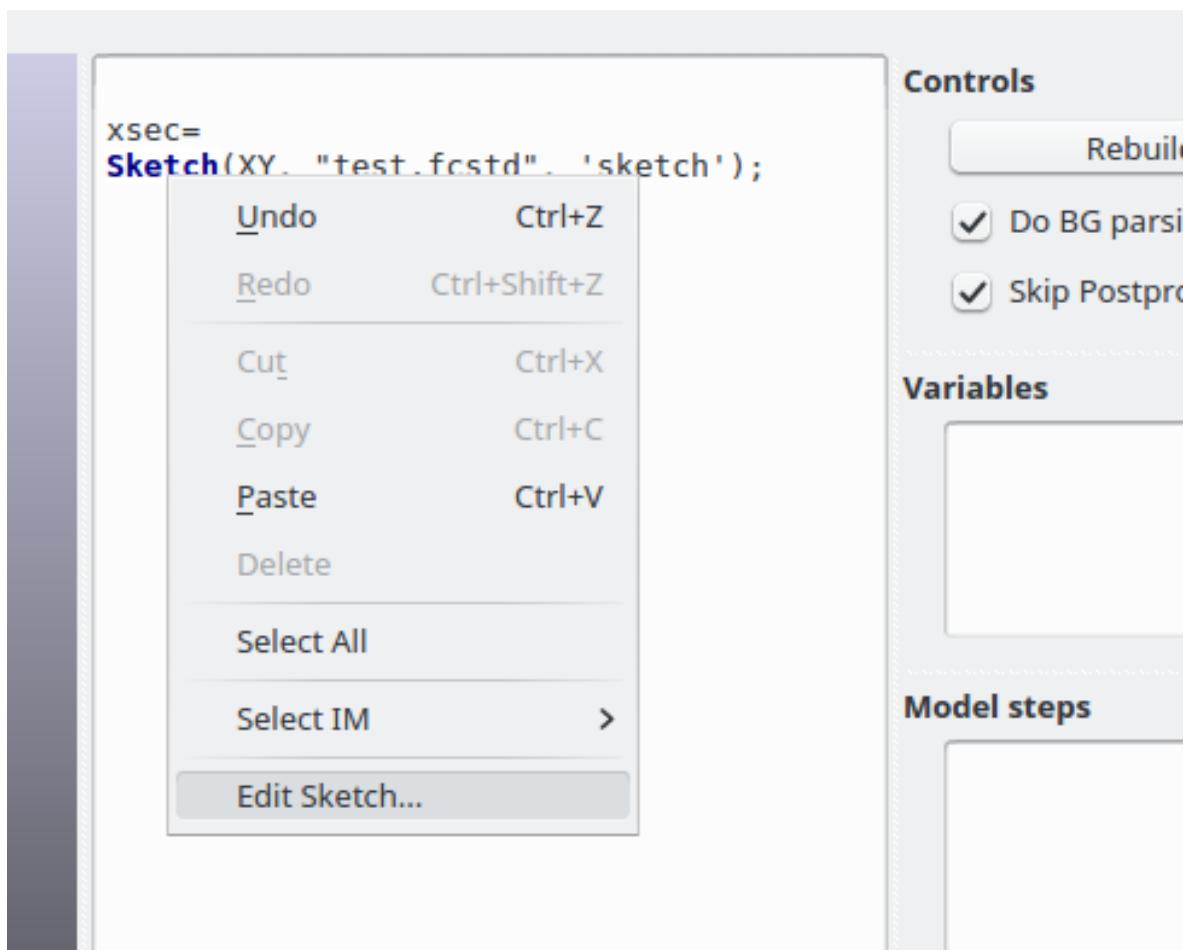
The context menu provides these functions:

- The first entry is the feature name. When it is selected, the definition in the script editor is highlighted and the cursor jumps to it.
- "Insert name": inserts the name of the feature symbol at the cursor location
- "Export...": Export the feature geometry to a file (BREP, STP, IGES, STL)

### 8.8.2 Text Editor

When script code is entered into the text editor window, it is parsed in the background. Once this has been done successfully, some extensions of the context menu is available:

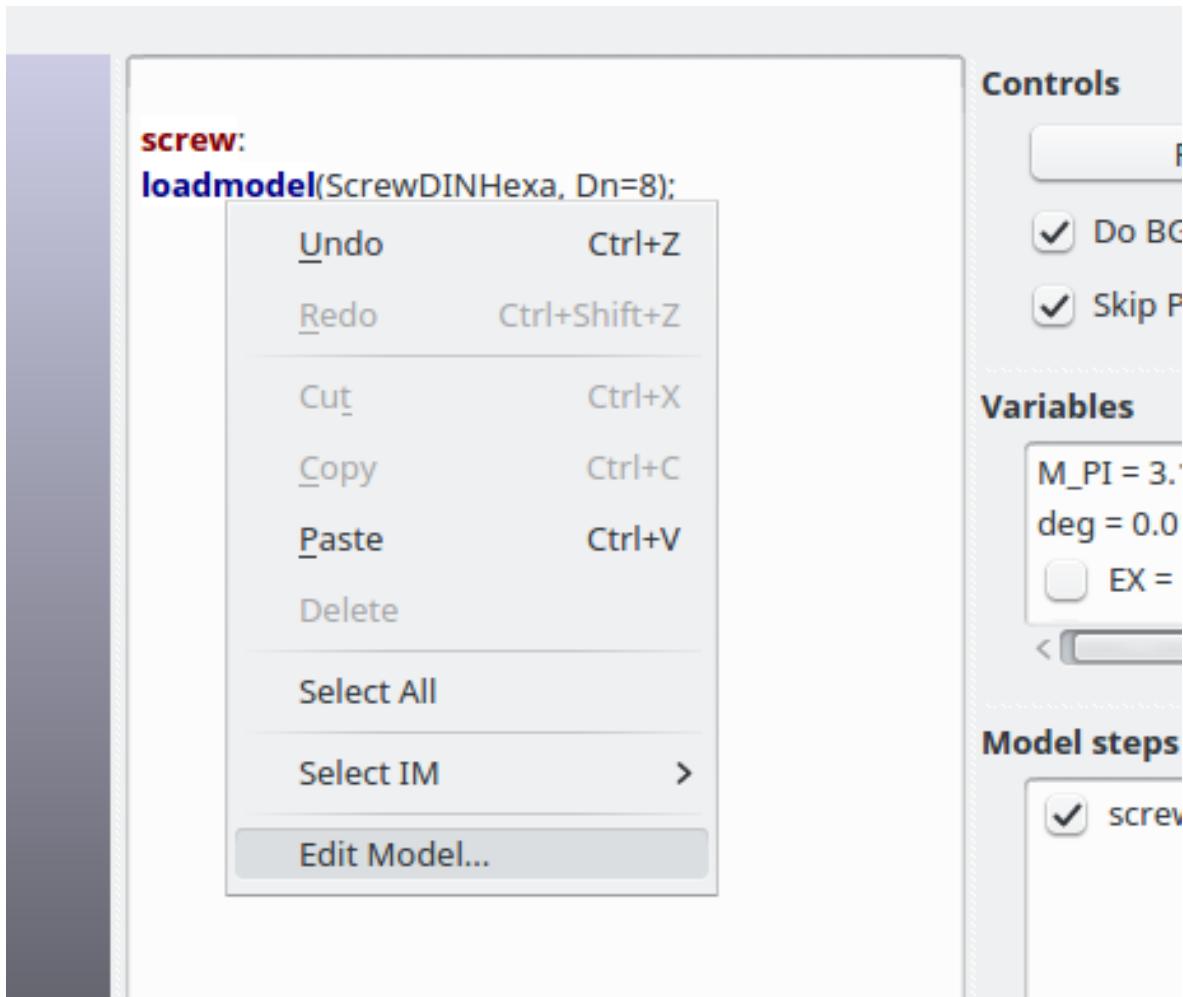
- Context menu on "Sketch" command:



When selecting the "Edit Sketch..." entry, FreeCAD is launched and the sketch editor opened. If the FreeCAD-file or sketch inside it is not yet existing, they are created.

- Context menu on "loadmodel" command:

<u>Project Codeword</u> doc	<u>Document No</u> 202002051012	<u>Editor</u>	<u>Date</u> November 24, 2021	<u>Revision</u> 0
--------------------------------	------------------------------------	---------------	----------------------------------	----------------------



When selecting the "Edit Model..." entry, another instance of iscad is launched with the specified model script loaded.

## 9 Developer Documentation

InsightCAE is a toolbox for the creation of custom workflows. It is therefore very natural to use its API in own extensions.

There are different developer perspectives:

1. Creating extensions (own workflow modules).

To write own workflow modules, just a plain binary installation of InsightCAE is required. It brings header files for its libraries and a CMake configuration which enables the developer to build his own extensions. It is recommended to create a custom library containing the additions (plug-in). See section 9.1 for a guide.

2. Modifying and extending the InsightCAE base project itself.

Since InsightCAE is free and open software, it is perfectly possible to take it and apply extensions or corrections or any other modification or improvement. Please note that silentdynamics GmbH develops InsightCAE primarily with its own applications in mind so there are almost certainly features missing in the API which third party users might need. If you

<u>Project Codeword</u>	<u>Document No</u>	<u>Editor</u>	<u>Date</u>	<u>Revision</u>
doc	202002051012		November 24, 2021	0

implement something which might be of public interest, please think about sharing it with us and the rest of the community. We are very open to integrate foreign contributions into the project.

Now, modifying the InsightCAE project requires to build the project from its sources. Please see section 9.2 for a guide to set up a self-built working copy of the sources.

## 9.1 Plug-In Development

## 9.2 InsightCAE Development

### 9.2.1 Dependencies

The InsightCAE toolkit requires a number of dependencies. Not all versions have been tested, of course. The following list contains the major dependencies along with the versions, which are currently utilized in binary packages:

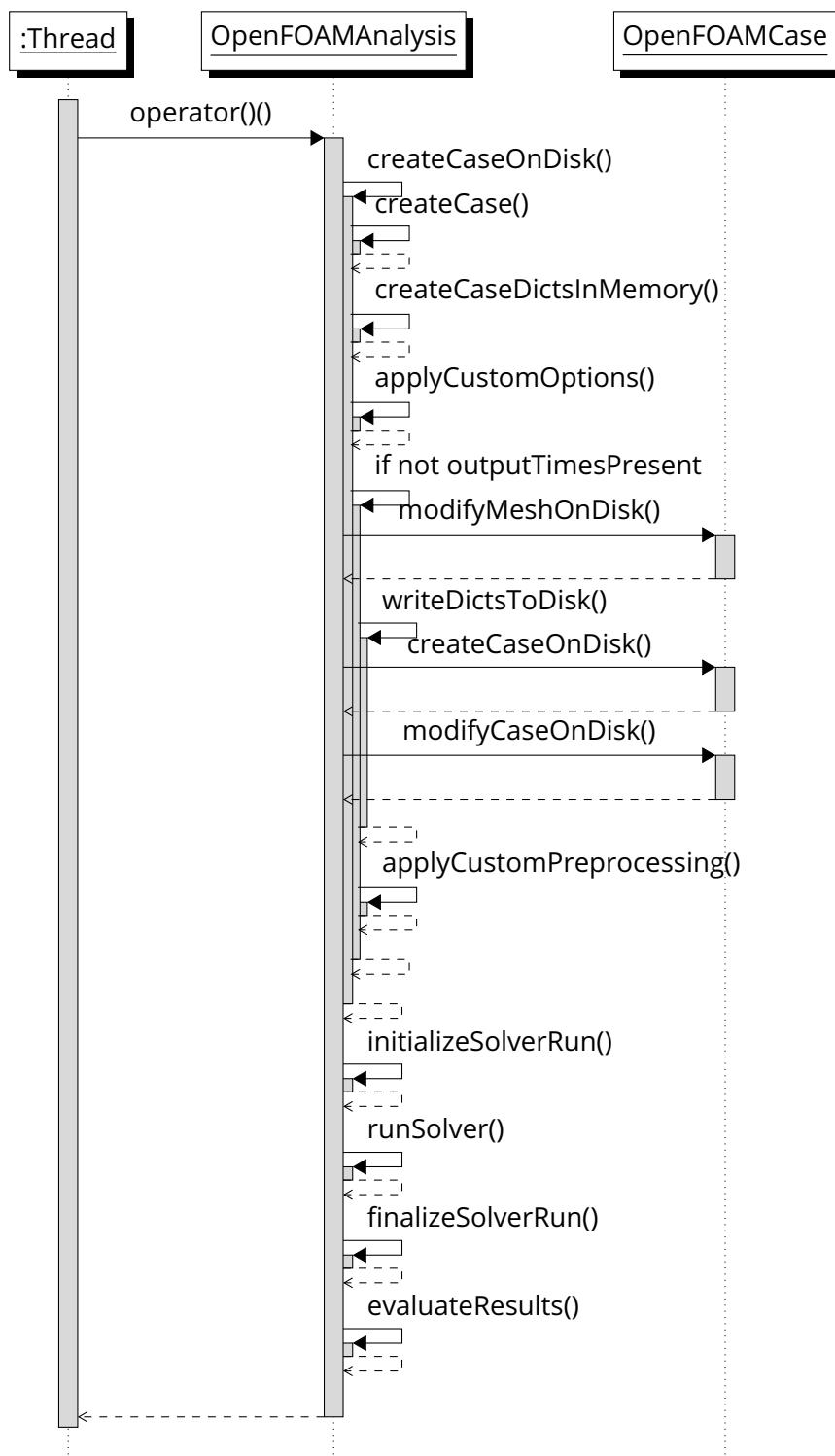
- boost, <http://www.boost.org>, 1.65.0
- python, 3.6
- gsl
- armadillo
- OpenCASCADE
- dxflib
- Gmsh
- Qt, 5.
- Wt
- OpenFOAM
- Code\_Aster

### 9.2.2 OpenFOAM Analysis

The file `openfoamanalysis.pdf` hasn't been created from `openfoamanalysis.dot` yet.  
Run '`dot -Tpdf -o openfoamanalysis.pdf openfoamanalysis.dot`' to create it.  
Or invoke `LATEX` with the `-shell-escape` option to have this done automatically.

Figure 8: Inheritance of OpenFOAMAnalysis class

Project Codeword doc	Document No 202002051012	Editor	Date November 24, 2021	Revision 0
-------------------------	-----------------------------	--------	---------------------------	---------------



## 10 Getting Started, Tutorials