

Unified Interface for CFD Softwares

1 — Last update: 2019/11/06

Table of Contents

1. Getting Started	1
1.1. Introduction	2
1.2. Softwares Used	3
1.3. Obtaining Gmsh	4
1.4. Obtaining Fluidity	5
1.5. Obtaining Paraview	6
1.6. Cloning Repository	7
2. Understanding Graphical User Interface	9
2.1. Setting up and Running Interface	10
2.2. Create New Project	11
2.3. Open Existing Project	13
2.4. Other Functionality	14
3. Pre-Processing	16
3.1. Creating a .Geo	17
3.2. Creating a Mesh	18
4. Solving	19
4.1. Creating .vtu files	20
5. Post-Processing	21
5.1. Plot data	22
6. Credits	23

1. Getting Started

1.1. Introduction

This software is Unified Interface for some pre-existing CFD softwares. Using this, user would not have to use different GUI's for different softwares and channel their output into each other, which can be very time consuming.

1.2. Softwares Used

Following Softwares are covered in this Unified-Interface

1. Gmsh
2. Fluidity
3. Paraview

These requires to be pre-installed in your system beforehand. To know how to obtain them, see the [Obtaining Gmsh](#), [Obtaining Fluidity](#), [Obtaining Paraview](#).

1.3. Obtaining Gmsh

To Obtain Gmsh in your UNIX system

```
* sudo apt-get install gmsh
```

In case this does not work

Uninstall the repository version.

```
* sudo apt-get remove gmsh
```

Download a zipped copy of the latest stable release for linux from the gmsh website: [Here](#)

Unzip gmsh:

```
* tar -xvzf gmsh-3.0.6-Linux64.tgz
```

(or whatever version you've downloaded)

This makes a folder gmsh-3.0.6-Linux64 containing gmsh which is portable. Put it somewhere you can find it. Navigate to the bin folder:

```
* cd ~/gmsh-3.0.6-Linux64/bin
```

From the bin folder you can start gmsh by typing:

```
* ./gmsh
```

To make it generally available at the command line, edit .bashrc :

```
* export PATH="$PATH:/home/bob/gmsh-3.0.6-Linux64/bin"
```

1.4. Obtaining Fluidity

To Obtain Fluidity in your UNIX system.

```
* sudo apt-add-repository -y ppa:fluidity-core/ppa  
sudo apt-get update
```

This assumes your local user account has sudo privileges.

- Installing Fluidity packages

To install a binary package of Fluidity which can be run from the command line, including the relevant supporting software, run:

```
* sudo apt-get -y install fluidity
```

1.5. Obtaining Paraview

In order to use scripting, Python is required (version 2.7.13 is known to work, as well as python 3.6. Make sure to install the right version for your compiler)

To Obtain Paraview in your UNIX system

✿ `sudo apt-get install paraview`

You must have setup the environment variable PYTHONPATH to the indicated initial value to avoid some missing packages when using Programmable Filters.

You must add this to your .bashrc to run anything in a ProgrammableFilter.

✿ `export PYTHONPATH=$PYTHONPATH:/home/doriad/bin/ParaView/Utilities/
VTKPythonWrapping/site-packages #fixes "no module named paraview"
export PYTHONPATH=$PYTHONPATH:/home/doriad/bin/ParaView/bin #fixes "ImportError:
No module named libvtkCommonPython"`

Note that for older versions of ParaView the first line may need to be replaced by:

✿ `export PYTHONPATH=$PYTHONPATH:/home/doriad/bin/ParaView/Utilities/
VTKPythonWrapping`

1.6. Cloning Repository

- On GitHub, navigate to the main page of the repository.
- Under the repository name, click Clone or download.



- To clone the repository using HTTPS, under “Clone with HTTPS”, click . To clone the repository using an SSH key, including a certificate issued by your organization’s SSH certificate authority, click Use SSH, then click .

Clone with HTTPS ?

[Use SSH](#)

Use Git or checkout with SVN using the web URL.

`https://github.com/Vaibhavsaharan/Uni`



[Download ZIP](#)

- Open Terminal.
- Change the current working directory to the location where you want the cloned directory to be made.
- Type `git clone`, and then paste the URL you copied in Step 2.

* `$ git clone https://github.com/YOUR-USERNAME/YOUR-REPOSITORY`

- Press Enter. Your local clone will be created


```
* $ git clone https://github.com/YOUR-USERNAME/YOUR-REPOSITORY> Cloning into
`Spoon-Knife`...
> remote: Counting objects: 10, done.
> remote: Compressing objects: 100% (8/8), done.
> remove: Total 10 (delta 1), reused 10 (delta 1)
```

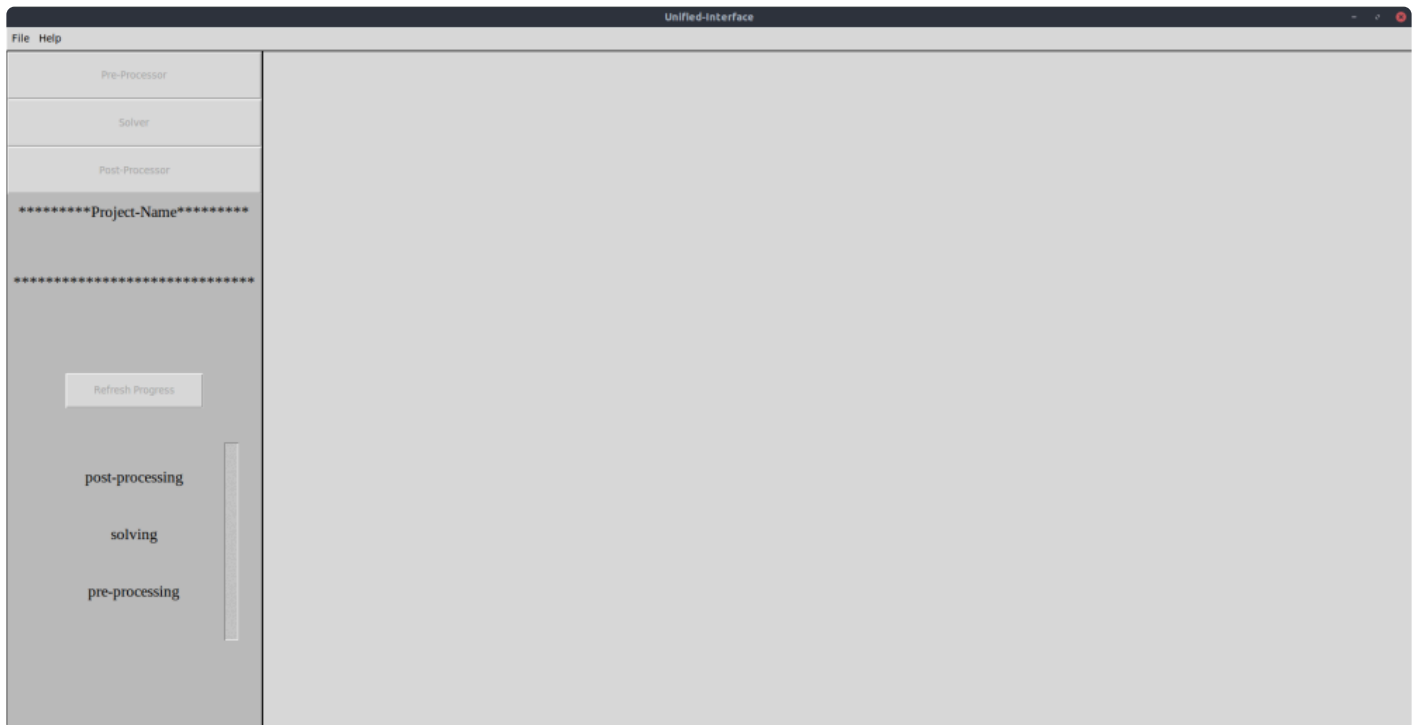
> Unpacking objects: 100% (10/10), done.

2. Understanding Graphical User Interface

2.1. Setting up and Running Interface

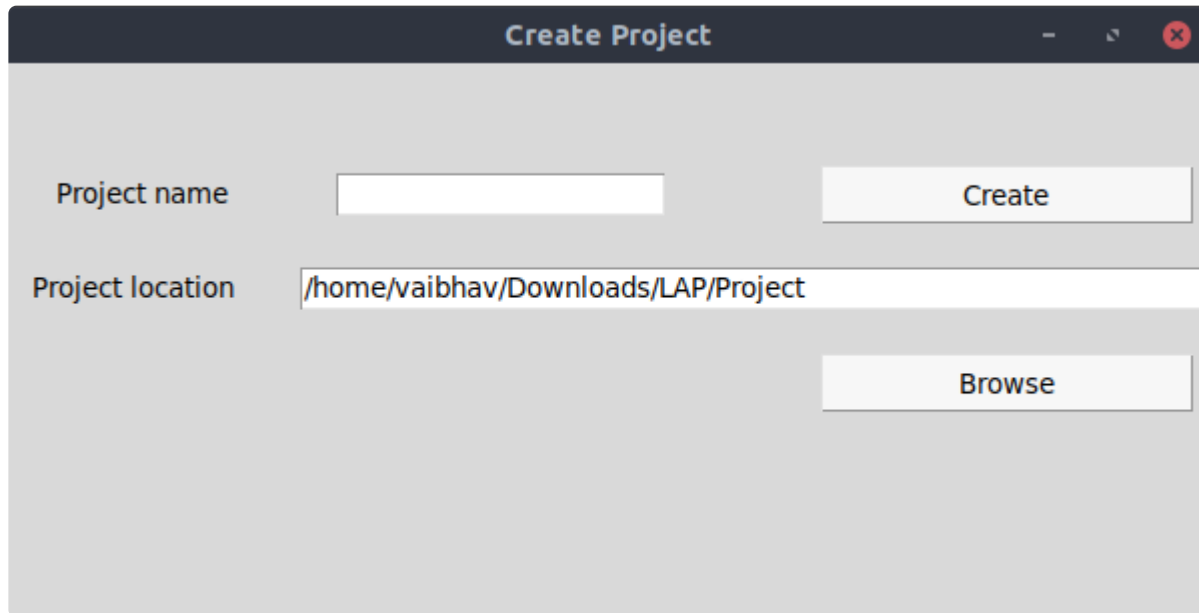
- Download the zip and extract contents in your working directory.
- Open terminal and go to your working directory
- Type the following command

 `python3 GUI.py`



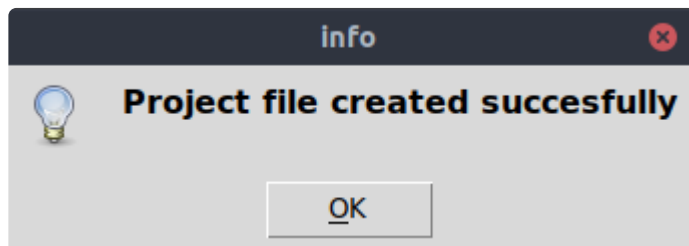
2.2. Create New Project


To Create a new Project, Go to **File** in menubar and click **New Project**. This will open a new window

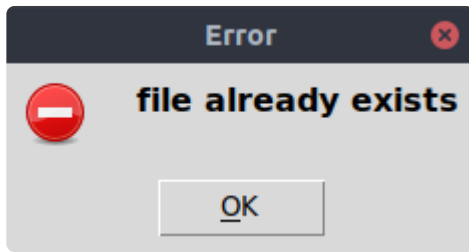


- Enter the project name in Project name entry and click create to make project folder in shown directory in project location.
- If you want to change the project location, click on browse button to choose directory location.

 Successfully created file will show this message

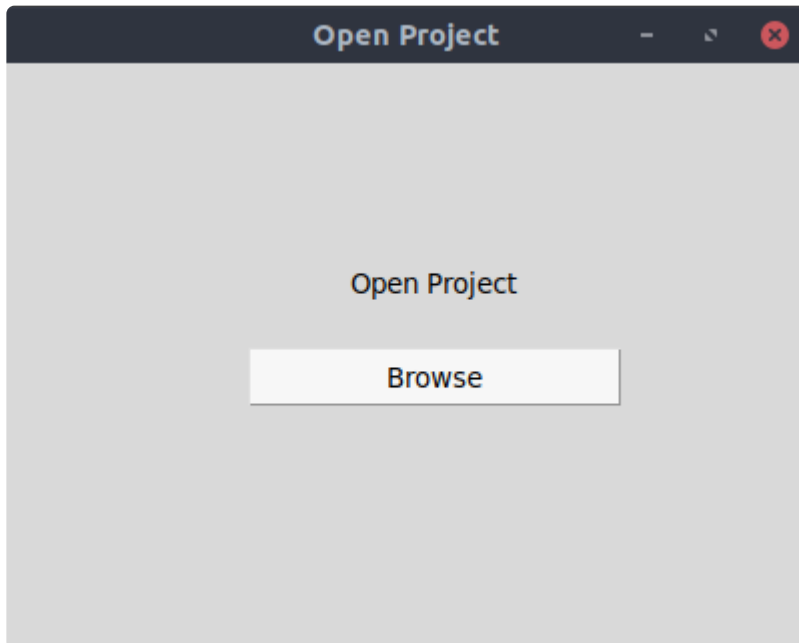


 Creating duplicate project will show this error



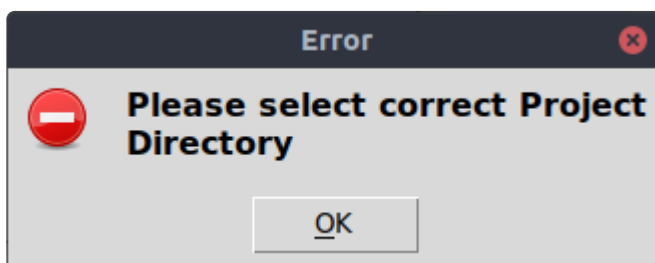
2.3. Open Existing Project

To Open an Existing Project, click on **File** in menubar and then click on **Open Project** which will open a new window.



- Click on browse button and double click on the project directory that you want to work on.

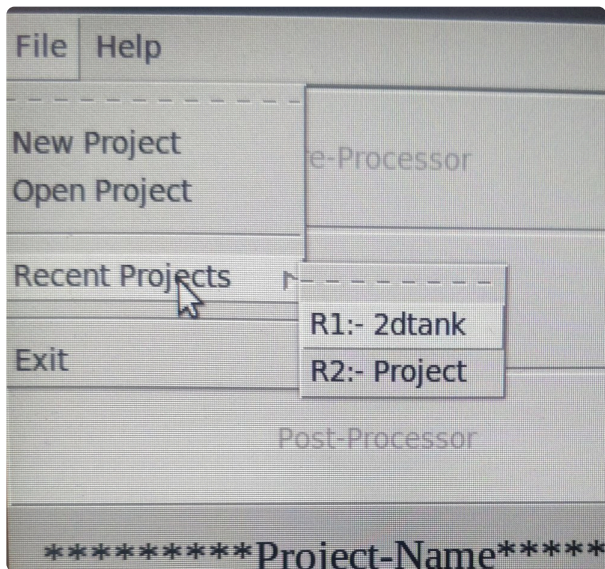
! Selecting directory that is not created by software will produce this error



2.4. Other Functionality

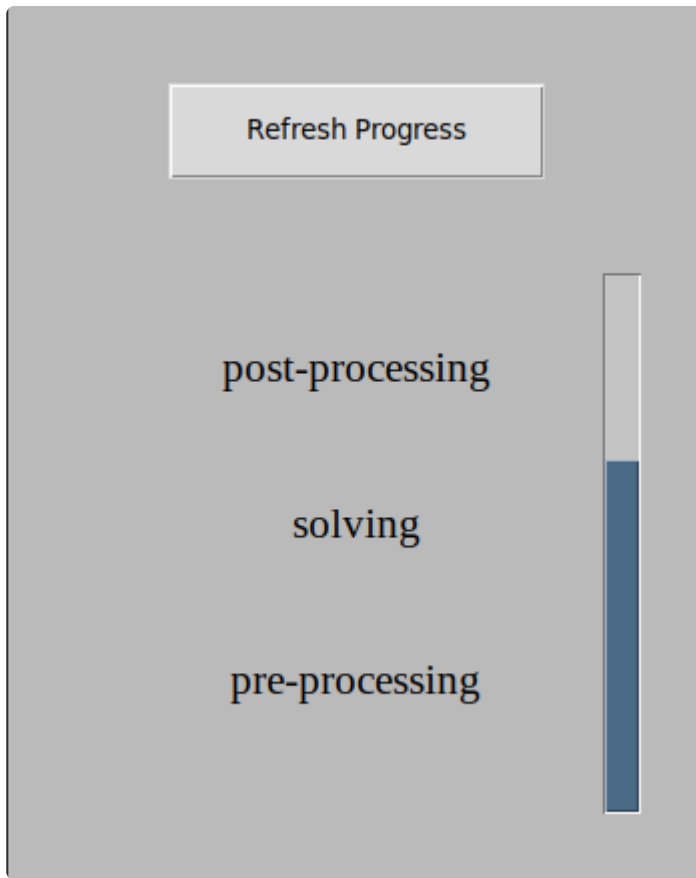
Recent Projects

- Choose between the projects that you have recently worked upon
- Increases the user experience and work speed.



Progress-Bar

- Progress-bar will show the real time project progress and one can resume further on the basis on this information



Project-Name

- Current working project name will be shown here

3. Pre-Processing

3.1. Creating a .Geo

Gmsh is a three-dimensional finite element mesh generator with a build-in CAD engine and post-processor. Its design goal is to provide a fast, light and user-friendly meshing tool with parametric input and advanced visualization capabilities.

Case – 1. .Geo file is not available in pre-processing directory

- Click on browse to browse a .geo file and it will get copied in pre-processing directory
- To Create a new .geo file from scratch, click on create button.

Geo file available	<input type="checkbox"/>	Choose your .geo file	<input type="button" value="Browse"/>	<input type="button" value="Create"/>
--------------------	--------------------------	-----------------------	---------------------------------------	---------------------------------------

Case – 2. .Geo file is available in pre-processing directory

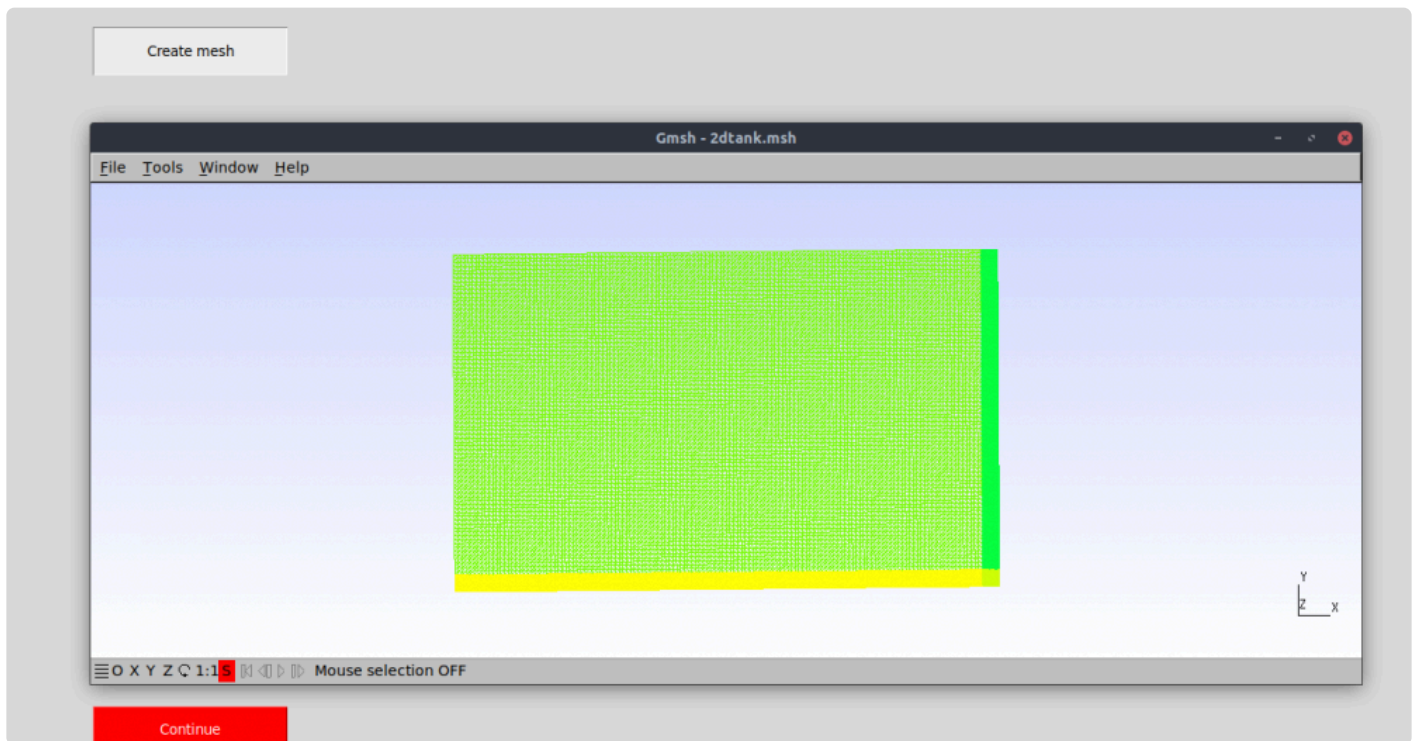
- File is automatically selected for mesh creating process.

Geo file available	<input checked="" type="checkbox"/>	2dtank.geo
--------------------	-------------------------------------	------------

3.2. Creating a Mesh

After the file is selected, Create msh button will become normal and you can create msh of your .geo file.

Click on Create Msh button to process into .msh file



Click on Continue button to progress further into Solving section

4. Solving

4.1. Creating .vtu files

Fluidity has very limited mesh generation capabilities, and in this context mesh-generation should not be confused with mesh adaptivity. However, mesh generation is an integral part of fluid-flow simulations.

After the file is selected, Solve button will become normal and you can solve your .flml file with given mesh.

This will store output files in post-processing folder.

5. Post-Processing

5.1. Plot data

ParaView is an open-source, multi-platform scientific data analysis and visualization tool that enables analysis and visualization of extremely large datasets. ParaView is both a general purpose, end-user application with a distributed architecture that can be seamlessly leveraged by your desktop or other remote parallel computing resources and an extensible framework with a collection of tools and libraries for various applications including scripting (using Python), web visualization (through ParaViewWeb), or in-situ analysis (with Catalyst).

Plotting of data is done by python scripting. Place your **plot_data.py** in same directory as **GUI.py**.

After the placement, click on Play button and python plots will become visible.

6. Creadits

Project Name

Unified Interface for CFD Softwares

Project Mentor

Dr. Gaurav Bhutani
Assistant Professor
School of Engineering
Indian Institute of Technology Mandi

Team Members

- Vaibhav Saharan
- Dinbandhu
- Devesh Soni