The Durham Pointwise Tutorial

Introduction

This provides a short step by step guide to generating a mesh in the Pointwise package. The aim being to get you started on the road to meshing and understand where to look for more detailed help. This tutorial is for Pointwise V17.1 Release 4.

Starting the Programs.

The fluid mechanics software packages are currently available on the CIS Linux service. Detailed instructions on how to access this service are available on this website:
https://www.dur.ac.uk/cis/linux/service_facilities/

All NPCS machines in the Engineering and Computing Sciences building are "dual boot". To access the Linux Desktop System (LDS) you will need to restart the machine and select "Linux" to run a linux system which will give you access to the fluid mechanics software. The remotely accessible service "mira" has the same versions of the software and can be accessed remotely.

To start up pointwise:

● Obtain a terminal window. Usually Applications → System Tools → MATE Terminal

● Start pointwise by typing *pointwise* at the terminal and then pressing enter.

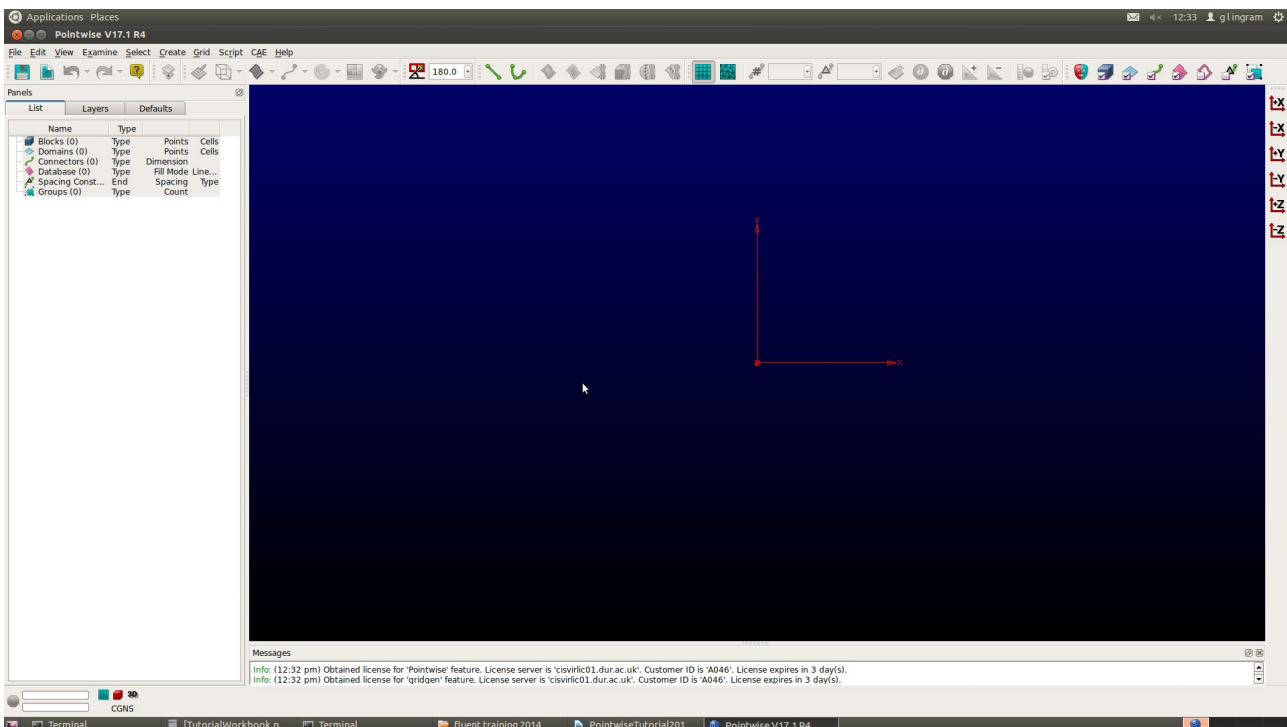● After a short while something like Figure 1 will pop up.

Figure 1: General View of Pointwise

The Help System

Pointwise features truly excellent documentation. The Help menu (top right menu) contains links to different types of documents.
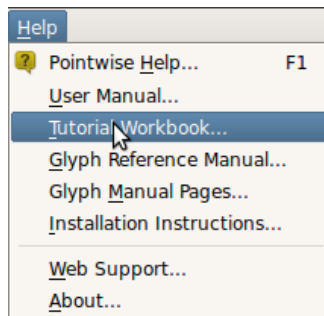

Figure 2: Help Menu

To get started the best document is the Tutorial Workbook. Help → Tutorial Workbook. This has step by step guides to a number of different problem types. Many of these require CAD files. These can be found in the tutorials directory: /opt/local/pointwise/current/tutorials

To copy the appropriate files to your local workspace, you can use the command line in a terminal to copy them across or navigate using the file manager. Places → Home Folder and then press Ctrl and L at the same time. This pops up a location bar into which you can type: /opt/local/pointwise/current/tutorials and then you have a file manager view of the files to copy.

Meshing a 2D Aerofoil

To do this we will follow the first tutorial in the Pointwise tutorial guide: "2D NACA 6412 Airfoil ". Note that since Pointwise is developed in the US spelling Airfoil is used.

- Firstly copy the files to your working directory:

  cp /opt/local/pointwise/current/tutorials/2DAirfoil/NACA6412.igs ./

- Help → Tutorial Workbook and find the 2D Airfoil tutorial. Then follow the instructions in the guide until you are ready to generate a mesh.

- Note that you quite often have to press toolbar buttons an index of these is given at the start of the tutorial. You might need to refer back to these as you complete the tutorial.

At the end of the tutorial you should have something that looks like Figure 3.
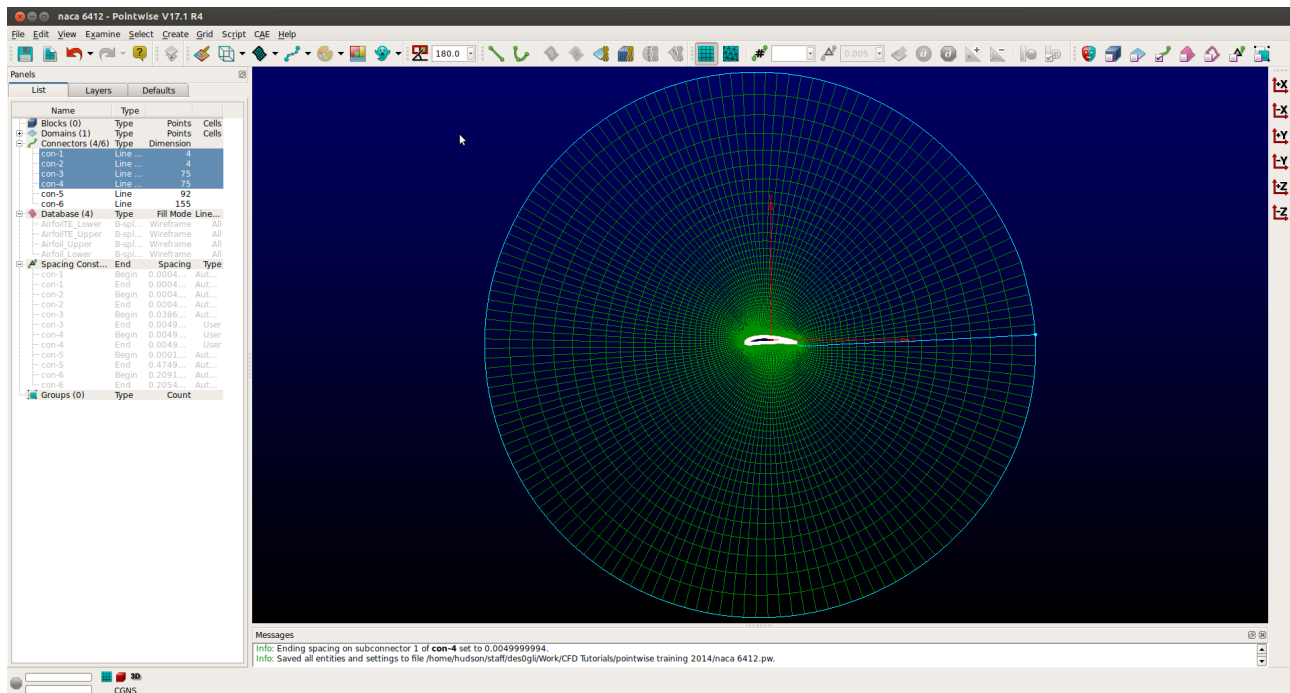
Figure 3: View of Pointwise After Finishing the Tutorial

We now need to set up Pointwise so that we can export the mesh.

- CAE → Select Solver. Then pick ANSYS Fluent.

- CAE → Set Dimensions → 2D.

- CAE → Set Boundary Conditions. This is a multi-step process.

  - You can use the Mask/Unmask tool so that only connectors will be picked up by the mouse

  - Click New

  - Click on the boundary you wish to set.

  - Having selected the boundary you wish to set up: set the type and name and make sure you click "Select" - the check box on the left hand side.

  - You need a minimum of two boundary conditions: one pressure far-field and one wall type for the aerofoil surface. Note that the aerofoil is made up of four connectors with two small ones at the trailing edge.

- CAE → Set Volume Conditions. New to setup new volume conditions in the same way as for the boundary condition.

- Use the List tab to select the domain (there should be only one!).

- File → Export → CAE to export a Fluent case file. (If you get problems when the mesh is imported into Fluent check the troubleshooting section).

Hints and Tips

You can use the examine menu to look at mesh quality. A high quality mesh is essential for a high quality fluid dynamics solution.

If you find menus greyed out and not available it means that you have dialogue open that Pointwise wants you to do something with. This is normally highlighted in red text on the top left of the application window.

At L1 you had detailed step by step courses on bits of engineering software (e.g. Solidworks). At L3 or L4 it is my expectation that you have the skills to learn how to use engineering software yourself so you should make extensive use of the help functions and the rich variety of on-line resources for the various fluid mechanics packages.

Pointwise has a Youtube channel featuring lots of hints and tips:
https://www.youtube.com/user/CFDMeshing/

Troubleshooting

Problem: When Pointwise starts it demands a license server.

Solution: This arises if you are using a local installation on a windows machine. Enter cisvirlic01.dur.ac.uk into the box and the program should start properly.

Problem: When using 2D meshes Fluent complains about negative volumes or can't read the mesh

Solution: You need to flip the orientation of the mesh. More details are here: http://www.cfd-online.com/Forums/pointwise/95388-pointwise-fluent-problem.html But the quick fix is to Edit → Orient and then swap I-J. This will flip the z-direction and keep Fluent happy - only do this if it doesn't work the first time.

Grant Ingram

5 November 2015