Anise GUI Tutorial

Artur Fast
Frederik Lührs
Mehrad Mohammadian
Tobias Lippert

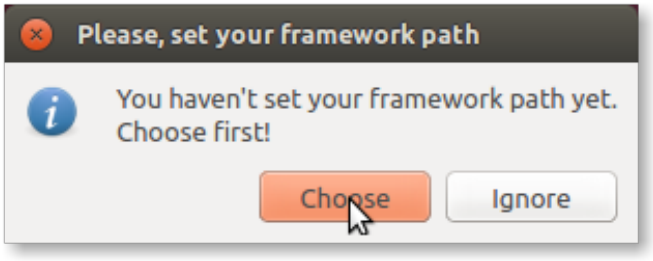
TECHNISCHE
UNIVERSITÄT
DARMSTADT

Department of Computer Science
Bachelor Lab WS14/15
March 31, 2015

This tutorial is to show you how to work with the *Anise GUI*. It is all about creating networks (meshes) of nodes and simulating your results with the *Anise Framework*.

Getting started

When you start the GUI for the first time you will have to select a Anise executable from your file system. You can skip this by clicking *Ignore*. This is not recommended because you will not be able to create meshes without a framework executable.
You can change the executable later on ("*Properties*" –> "*Select Framework Executable*").



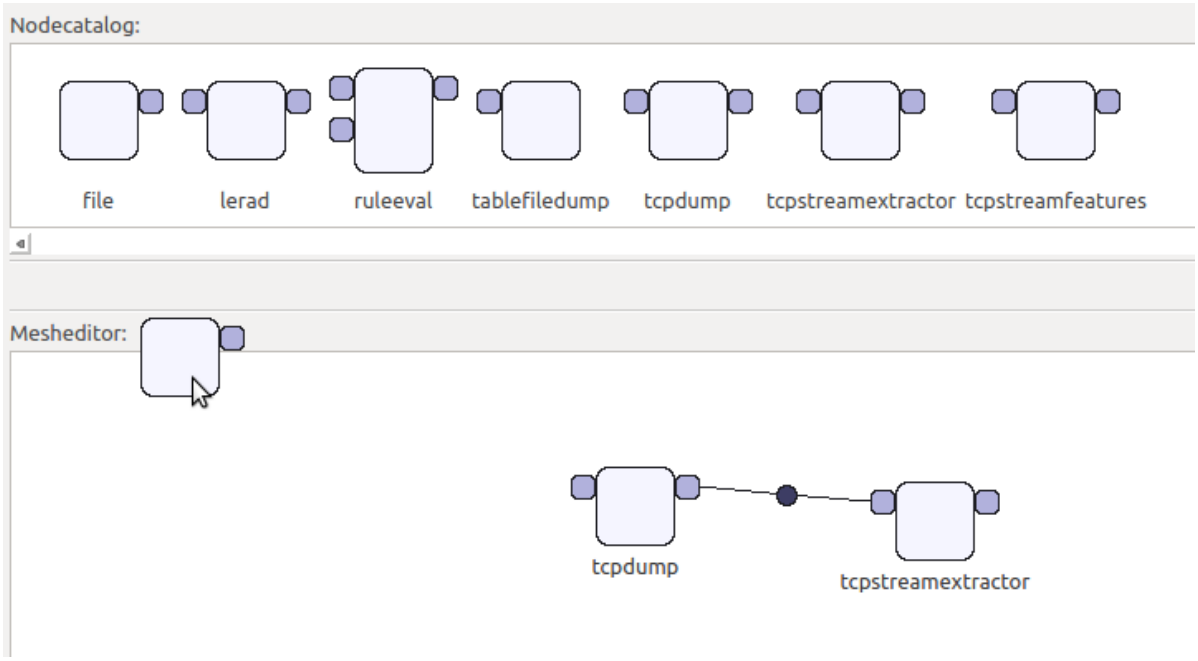
The GUI has to restart afterwards.

Loading a file

If have previously saved or self-written JSON files you want to load to the GUI you can do this by choosing "load" form the file menu. You can also use the short cut *CTRL + L*.

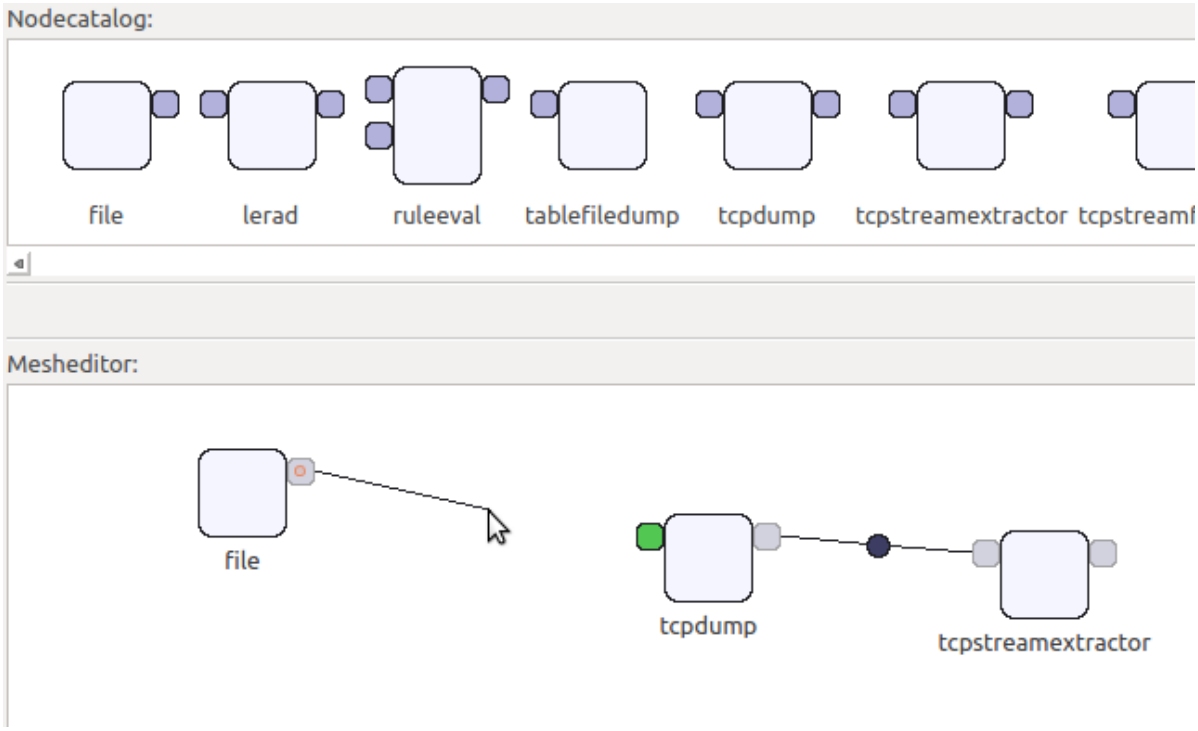
Create own meshes

You can create your own meshes by simply dragging nodes from the catalog on the top into the editing area in the center.



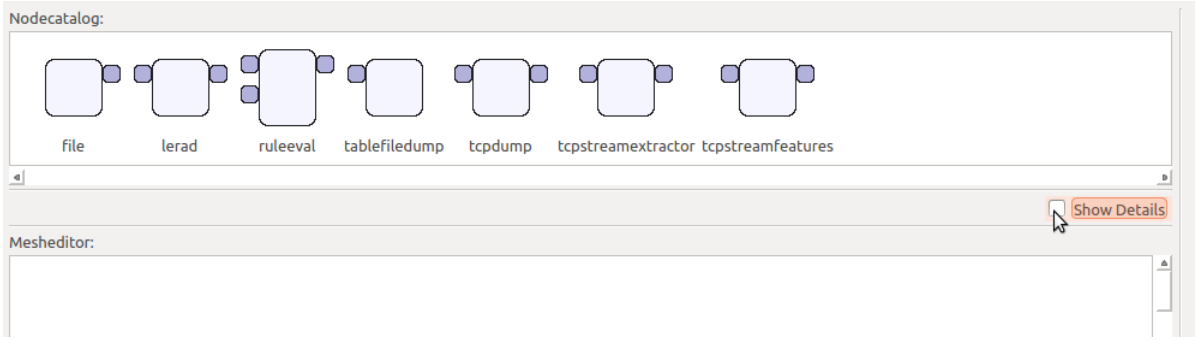
Adding connections

You can create connections between nodes by clicking on an output gate (right side of a node). You automatically enter the connection mode. All possible destinations you can draw a connection to will be highlighted in green.
If you want to add joints to your connection click on the spot the joint should be placed. To exit connection mode without creating a connection right-click anywhere.

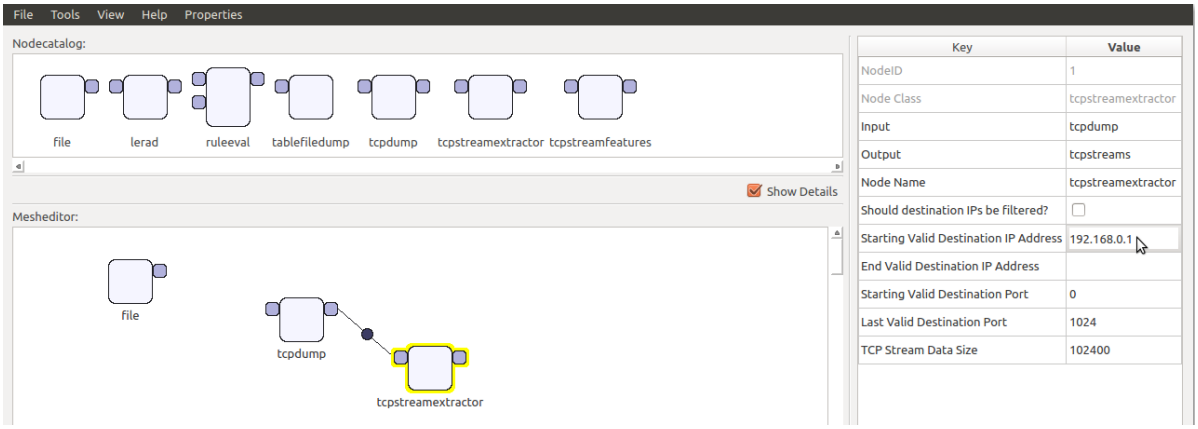


Editing node parameters

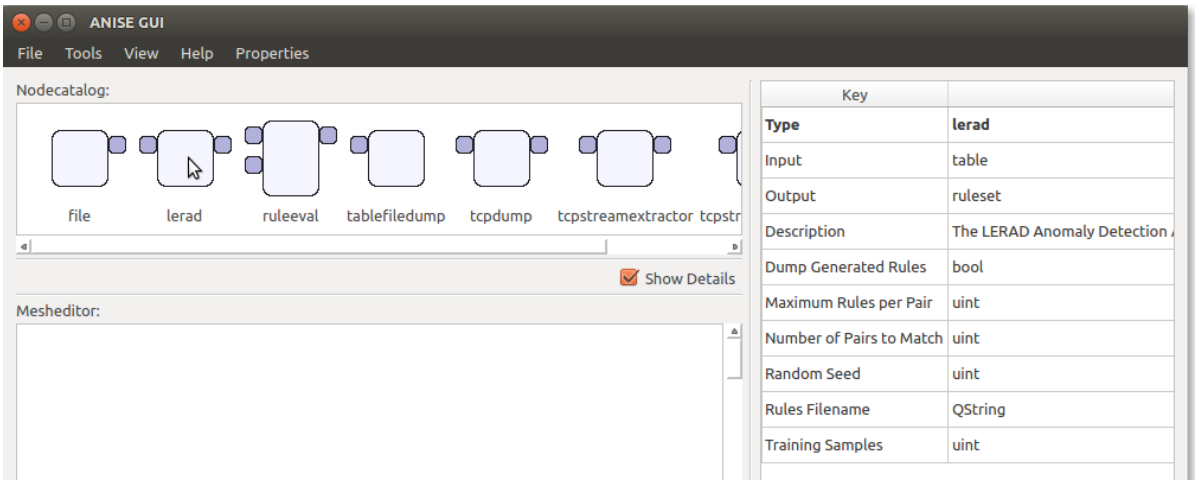
For editing node parameters you will have to enable the "*Show Details*" option in the top right corner.



When you now select a node, its description, name, parameters and further information will be shown on the right side. You can edit the parameters and the node's name.



If "*Show Details*" is checked, all relevant information about the nodes inside the catalog will also be shown on the right side by clicking on them.



Simulation

Before you execute your mesh you will have to save it. Start the simulation with "*Tools*" –> "*Start Simulation*" or use *CTRL + R*.

Every node has three states:

- **idle**
- **initializing**
- **procressing**