

MOSM – Design for Additive Manufacturing

Tutorial:

Lattice Modeling via Rhino GH Applications

Version 1.0
Spring 2024

Dr. Zhiping WANG

Lecturer at SeaTech, École d'ingénieurs de l'université de Toulon
zhiping.wang@univ-tln.fr

1. Introduction

Objective:

In this TP, we will learn how to do Lattice Configuration Optimization via Millipede plugin in GH. Millipede is a structural analysis and optimization plugin. Additionally, it exposes functionality that is relevant to the solution of many numerical and geometric problems.

Demonstration tool: Crystallon, Dendro and Millipede.

Millipede is a structural analysis and optimization component for grasshopper. It allows for very fast linear elastic analysis of frame and shell elements in 3d, 2d plate elements for in plane forces, and 3d volumetric elements. All systems can be optimized using built in topology optimization methods and have their results extracted and visualized in a variety of ways.

Octopus is originally made for Multi-Objective Evolutionary Optimization. It allows the search for many goals at once, producing a range of optimized trade-off solutions between the extremes of each goal.

Millipede: <https://drive.google.com/file/d/1revQ4mcTGKkUwyvQCteimUWrffa2AyW-/view?usp=sharing>

Octopus: <https://www.food4rhino.com/en/app/octopus>

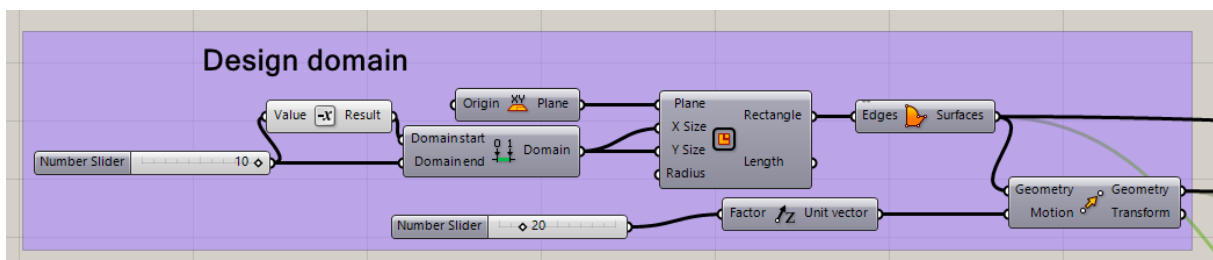
2. Tutoring example

In this tutoring example, we will optimize lattice structures configuration for a simple example.

2.1. Parametric model creation

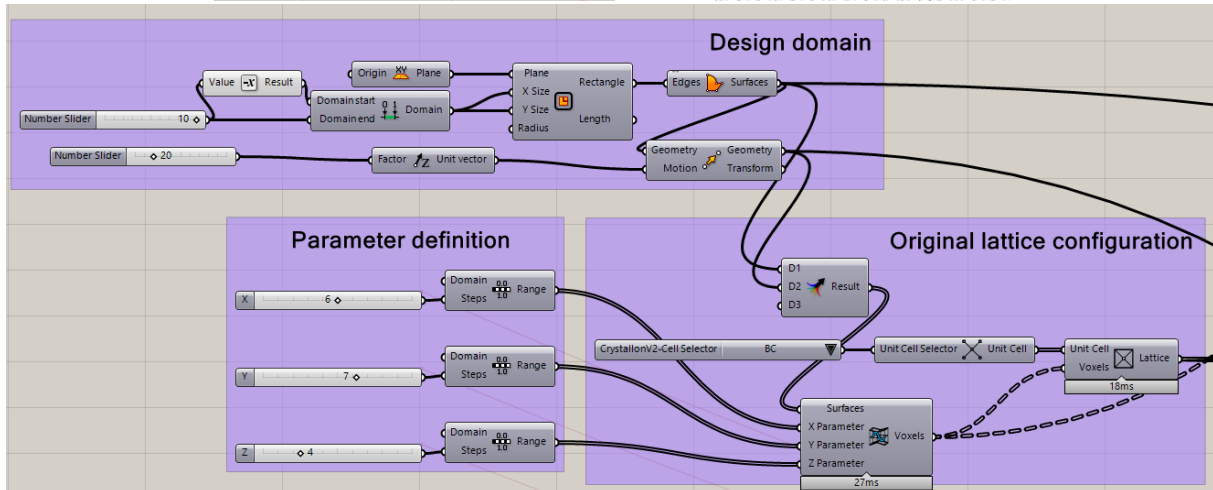
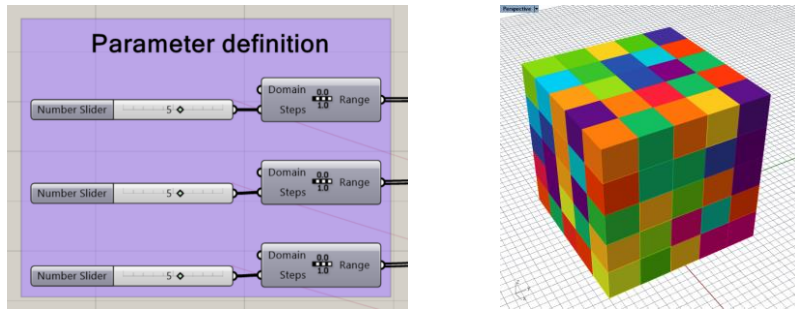
Step 1: Create two surfaces

Use **Rectangle** and **Boundary surfaces** components to generate a rectangle surface. Use **Move** component to move the surface along z direction.



Step 2: Generate parametric lattice configuration

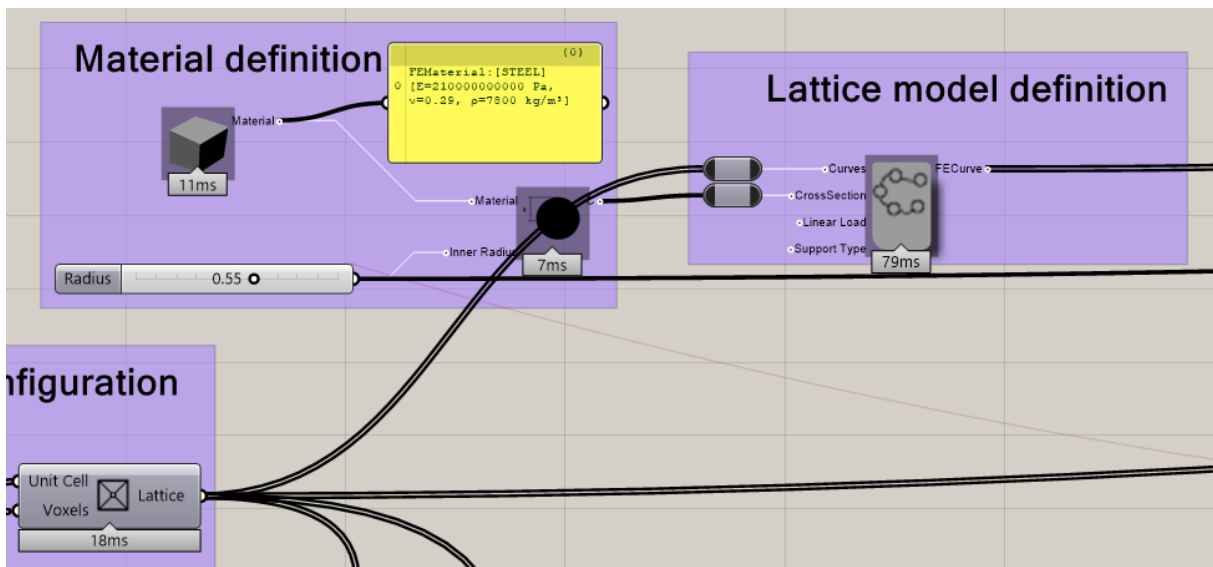
We use **Crystallon** component to generate parametric lattice configuration (More detail can be found in TP 3.)



2.2. FEA system

Step 1: Material definition

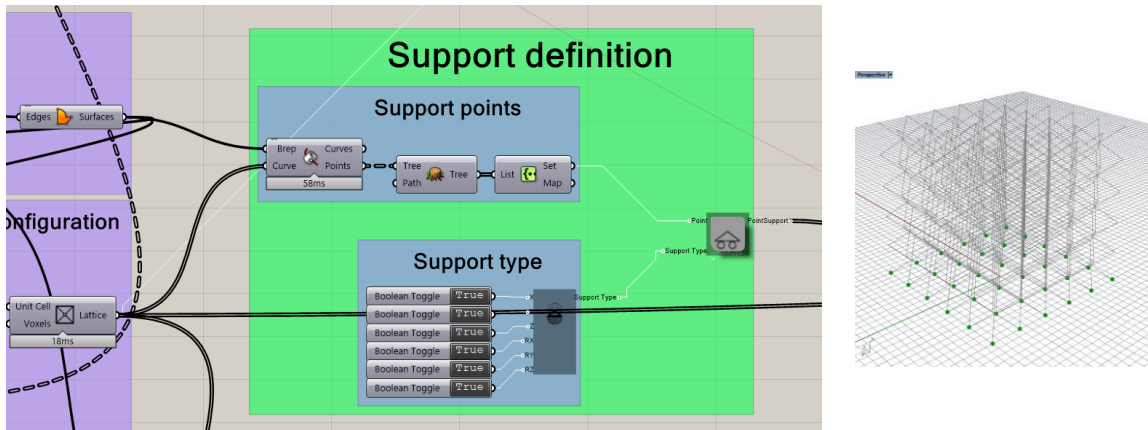
Drag **Material** component from **Millipede** plugin to the interface. Right click to choose **Steel**. Set radius as **0.55 mm** (range: 0.2 - 0.8mm). Connect **Material** and **Radius** number with **Cross Section** component. Right click to select **Circle solid**. Connect Lattice curves and Cross Section with **Frame curves** component.



Step 2: Support definition

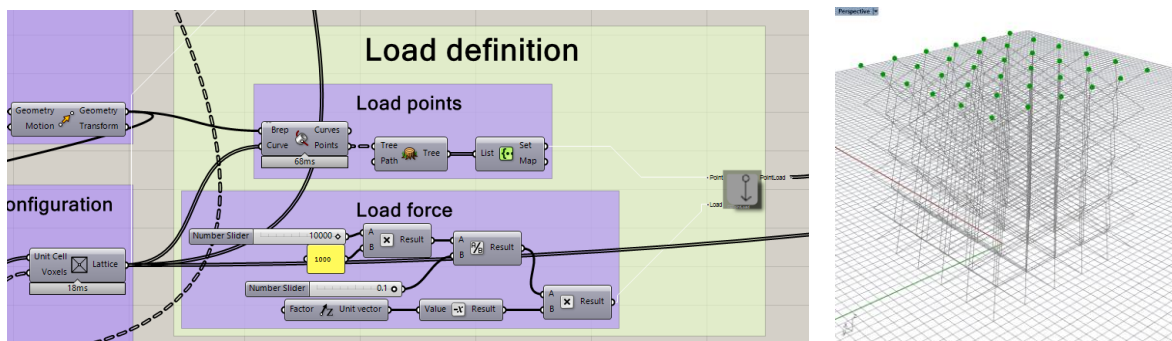
Use **Brep Curve** component to obtain support points. To delete repeat points, we flatten the intersection points and use **Create Set** component to select unique support points. Drag

Support type component from **Millipede**, set all input as **True**. Connect support points and the support types with **FE Point support** component.



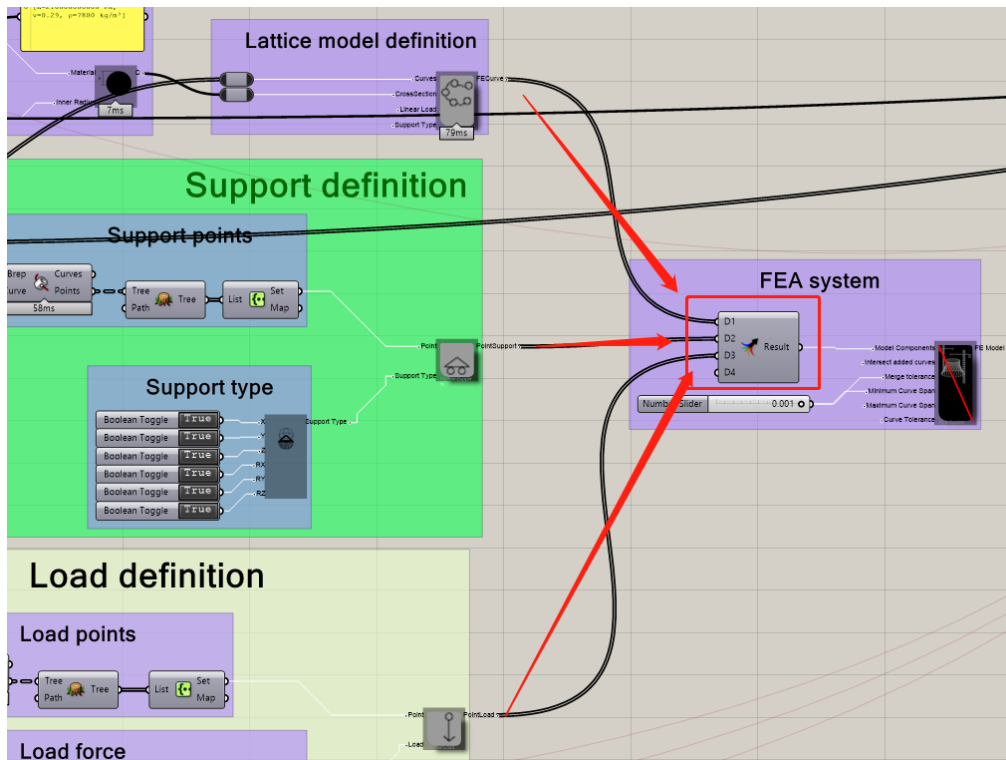
Step 3: Load definition

Use **Brep Curve** component to obtain load points. To delete repeat points, we flatten the intersection points and use **Create Set** component to select unique load points. Use different operators to generate a load force. Connect Load points and Load force with **FE Point Load** component.



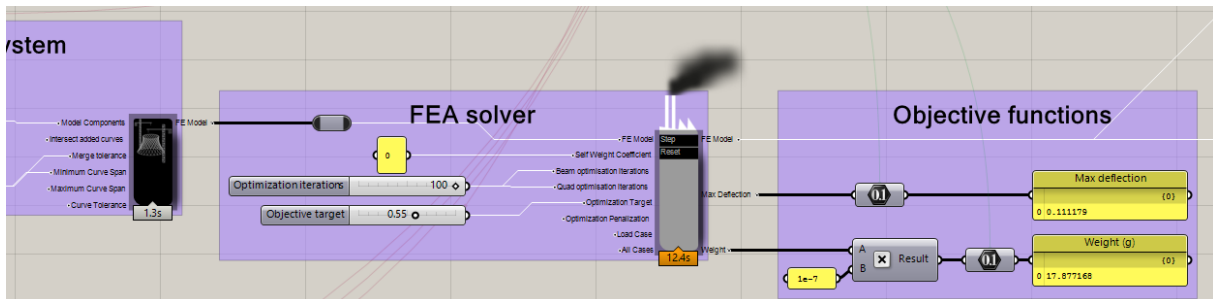
Step 4: FEA system

Use **Merge** component to connect lattice model definition, support definition and load definition to **FEA system** component. Set **Merge tolerance** and **Minimum curve span** as 0.001.



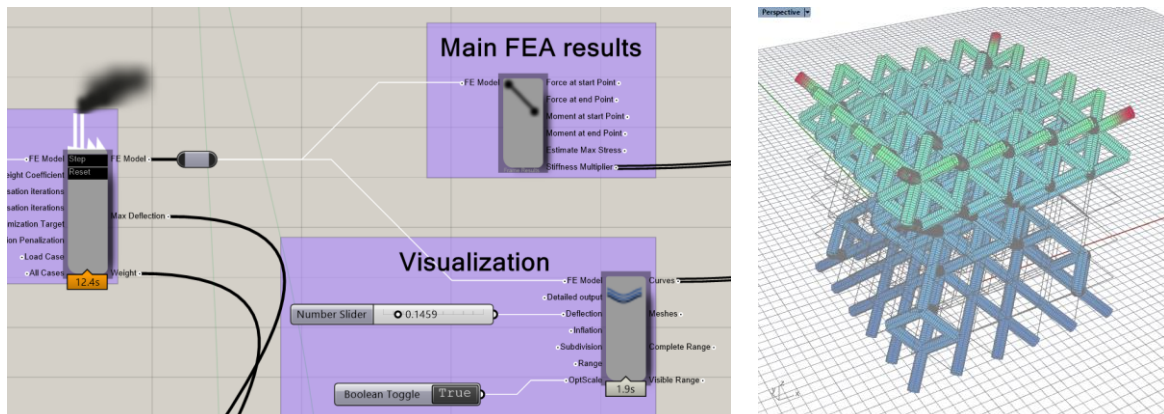
Step 5: FEA Solver

Link the **FE Model** with **FEA Solver** component. Set two optimization iterations and optimization target as 100 and 0.5, respectively. Two FEA results, **Max deflection** and **Weight**, are set as objective functions to optimize lattice structure configuration.



Step 6: FEA results visualization

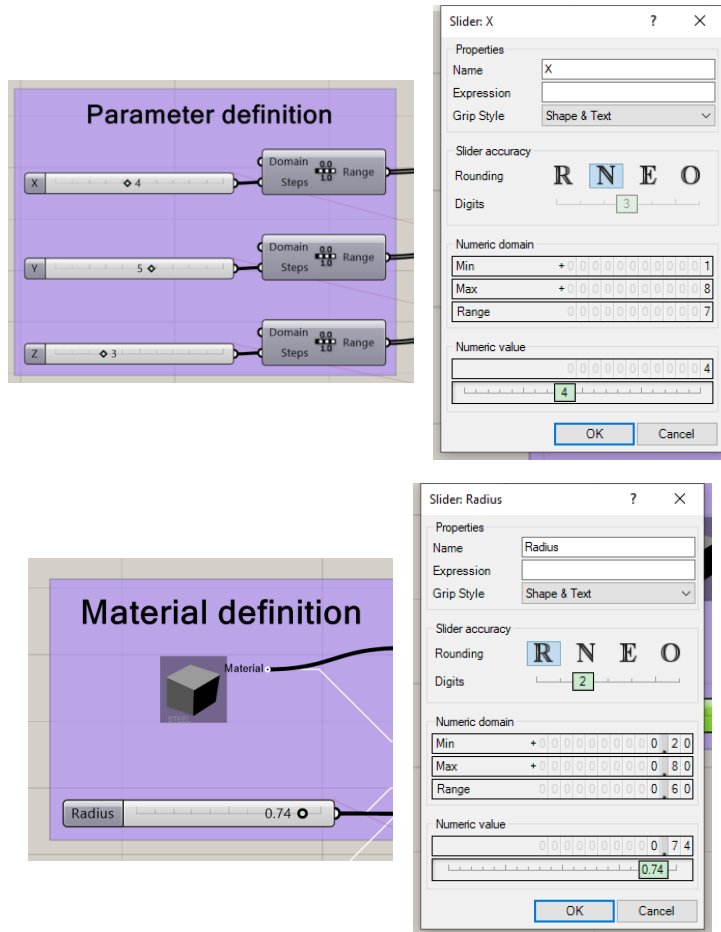
Connect **FEA solver** with **Frame results** and **Frame visualization** components.



2.3. Lattice configuration optimization

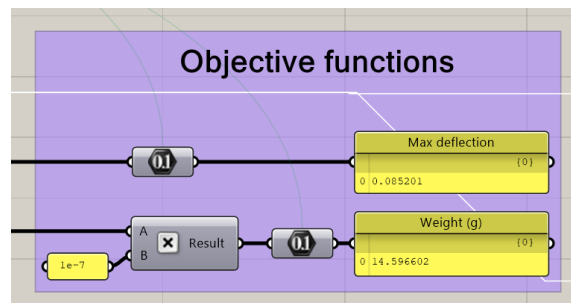
Step 1: Design variable definition

Three main parameters that control the number of lattice cells along x, y and z direction (range: 1 to 8), a maximum radius of lattice strut (range: 0.20 to 0.80 mm).



Step 2: Objective function definition

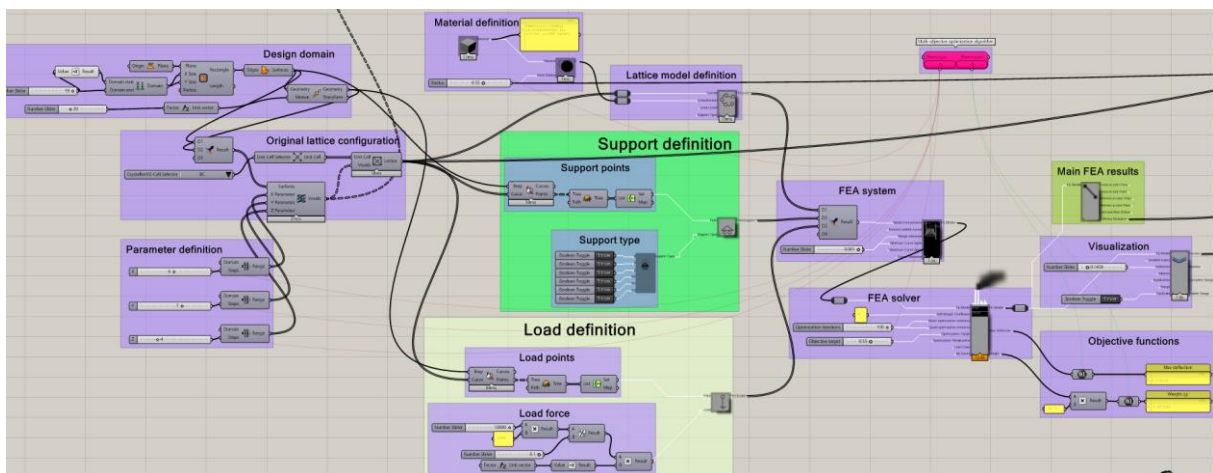
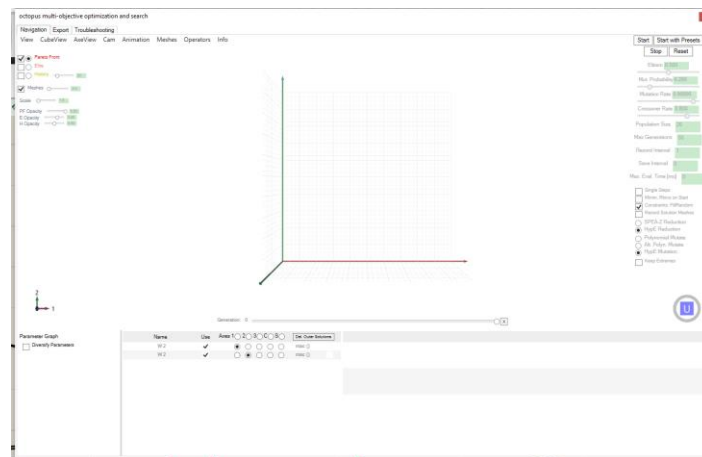
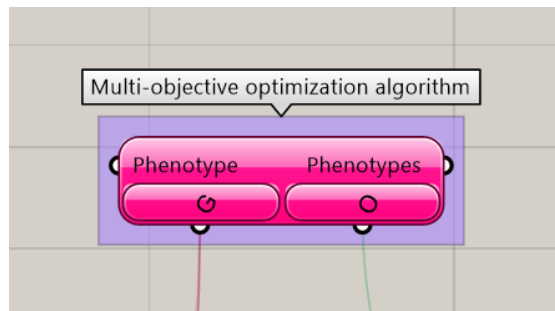
Two objective functions, **max deflection** and **weight**, are defined to optimize lattice configuration.



Step 3: MOEA solver

Drag **Octopus** optimizer to the interface. Connect design variables and functions with input. Two objective functions, max deflection and weight, are defined to optimize lattice configuration.

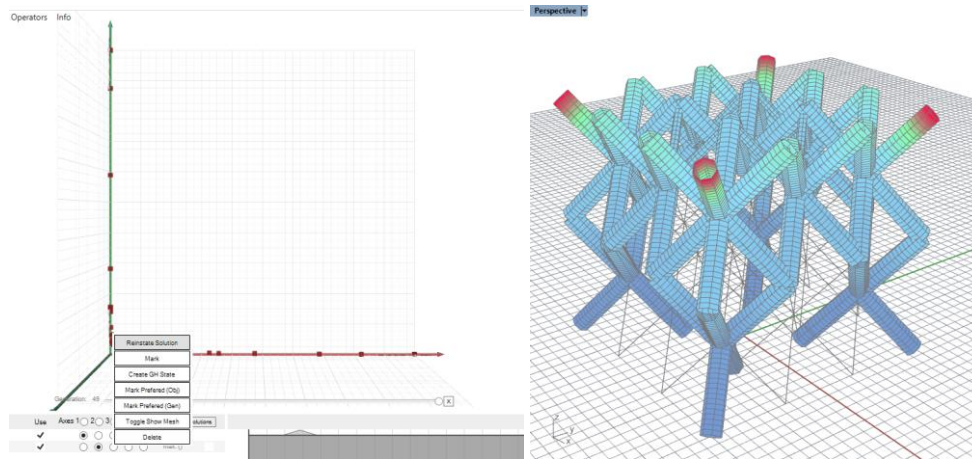
Double click **Octopus** component, set population size as 20, max generation as 50. Click start to start the MOEA solver.



2.4. STL file export

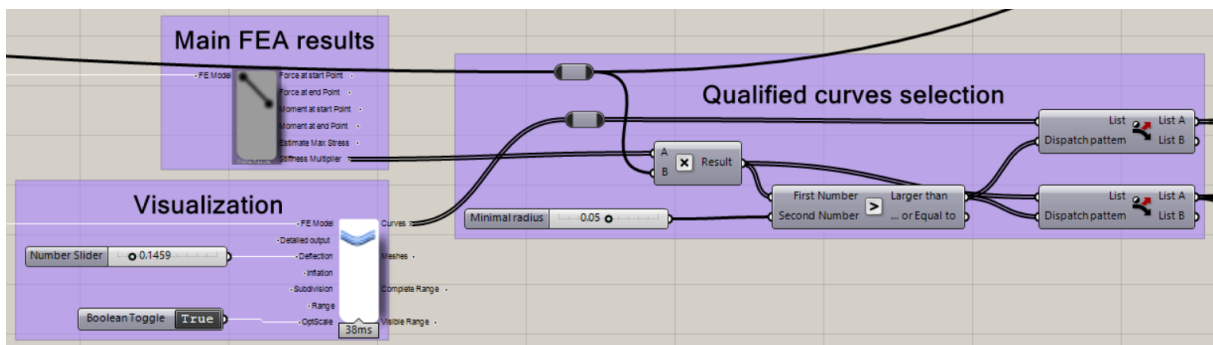
Step 1: MOEA result visualization

Reinstate one solution on the Pareto front.



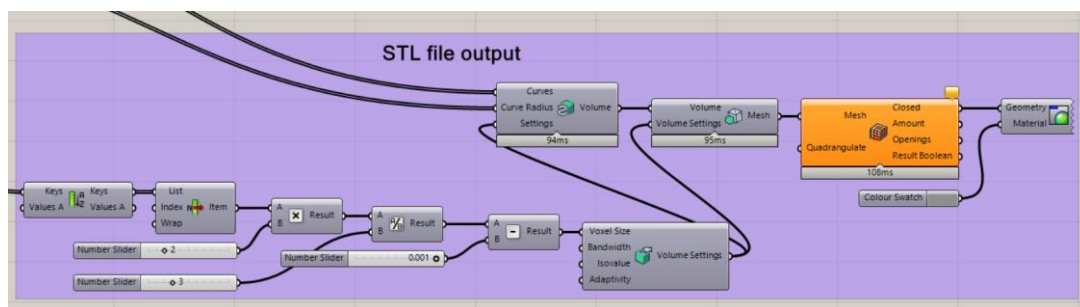
Step 2: Delete unqualified struts

Based on **stiffness multiplier** values, qualified struts are selected. The minimal value is set as **0.05**.

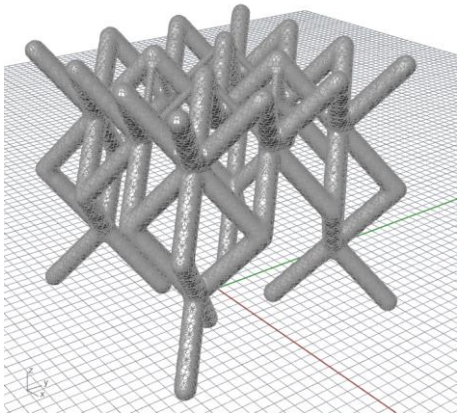


Step 3: STL file export

Use **Dendro** plugin to export STL files.



Perspective ▾



Perspective ▾

