MOSM – Design for Additive Manufacturing

Tutorial:

Parametric Design tool: Rhino GH

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 parametric design by using Grasshopper, which is a visual scripting platform that is included in Rhino. With Grasshopper, you can write scripts to automate tasks by dragging components to a main interface. Parameter control in Grasshopper can help you drive infinite design solutions that are previewed in Rhino without generating geometry. When final design solutions are selected, geometries are created by "baking" into the Rhino object. In this process, you can integrate objective functions and constraints within your design problem to find an optimal design. The Grasshopper environment provides an intuitive way to explore designs without having to learn to script. This will be quiet convenient for users to achieve parametric modelling for structural engineering.

Demonstration tool: Rhino Grasshopper (a visual programming language)

Grasshopper is a visual programming language and environment that runs within the Rhinoceros 3D computer-aided design application. In Rhino 8, Grasshopper is embedded into Rhino. To start Grasshopper, you can enter Grasshopper at the command-line and press Enter to activate the command.
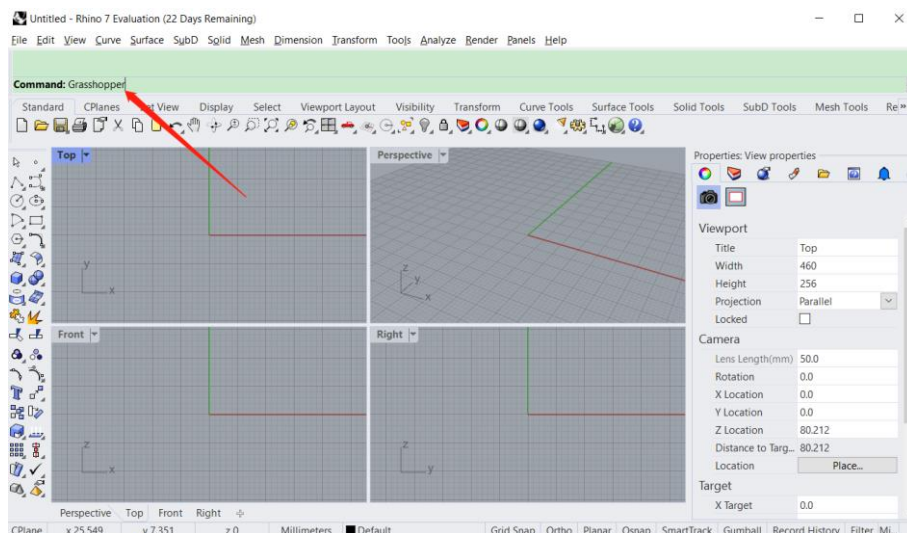

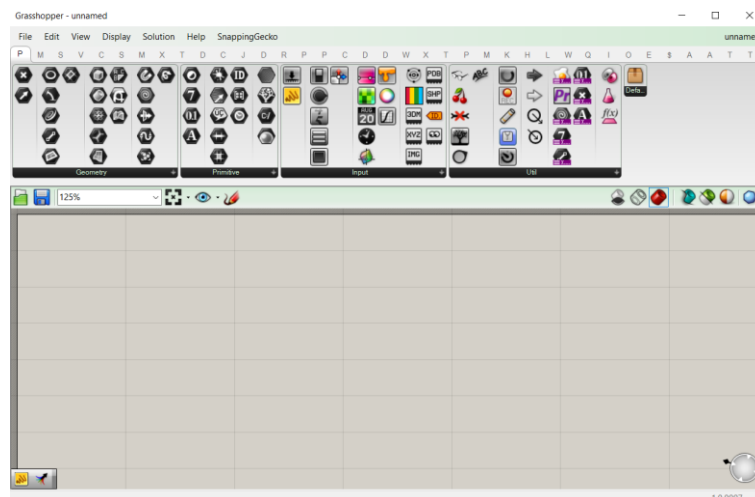Fig. 1. Rhino main interface and how to start Grasshopper.
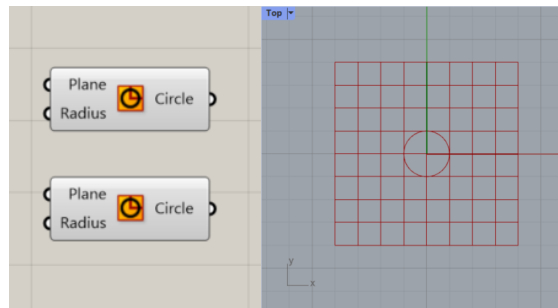
Fig. 2. Grasshopper main interface.

2. Tutoring example

In this tutoring example, we will design a bike wheel by using GH.

2.1. Create the Circles

Step 1: On the Grasshopper **File** menu, pick **New Document**.

Step 2: On the Grasshopper **Curve** tab, and the **Primitive** panel, drag and drop two **Circle** components on to the Grasshopper canvas.
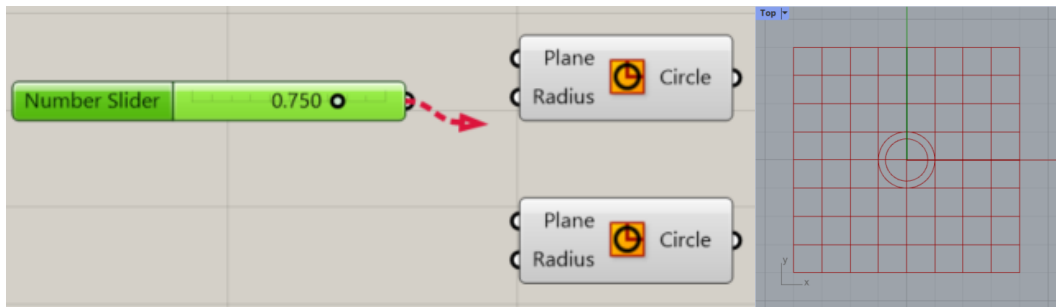

Step 3: Double-click on the Grasshopper canvas to open a dialog box with the prompt Enter a search keyword.

Step 4: Type **Number** and pick **Number slider** from the menu.

Step 5: A **Number slider** parameter will be added to the canvas.

Step 6: Drag the output connector from the **Number slider** to the input **Radius** of the Circle.


Step 7: Now drag the slider and see the radius of the circle in the top view update.

Step 8: To create the second **Number slider**, double-click on the canvas and type: 24<32<36. With this syntax, the Number slider will be set to 32, range from 24 to 36.

Step 9: Connect in the **Number slider** output to the input **Radius** on the second circle component.

Step 10: Double-click on the **Radius** label on the first number slider. The slider dialog will appear.

Step 11: Edit the **Min** and **Max** values. Set the **Min** to 1, the **Max** to 10, and the **Rounding** to N or natural number.
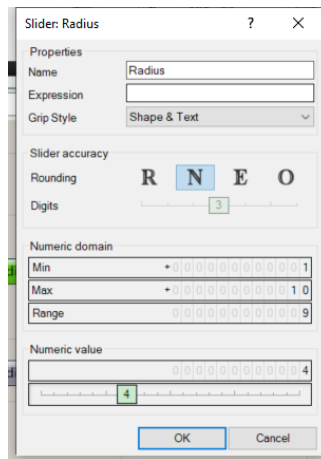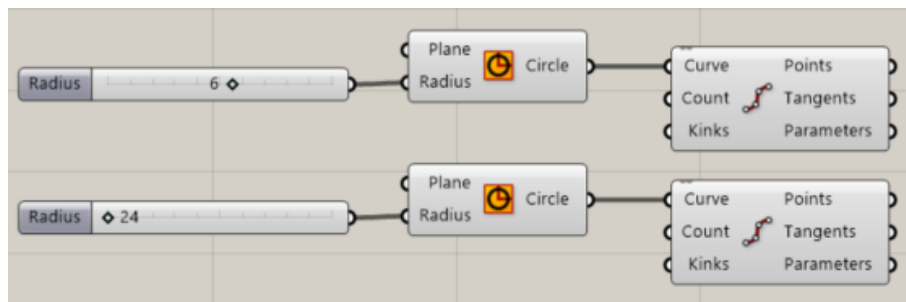
Step 12: Pick the OK button to close the dialog.

Step 13: Drag the first slider to 6.

2.2. Divide the Circle

Step 1: On the **Curve** tab, under **Division** panel, click **Divide Curve** and drop two on to the canvas to the right of the circles.
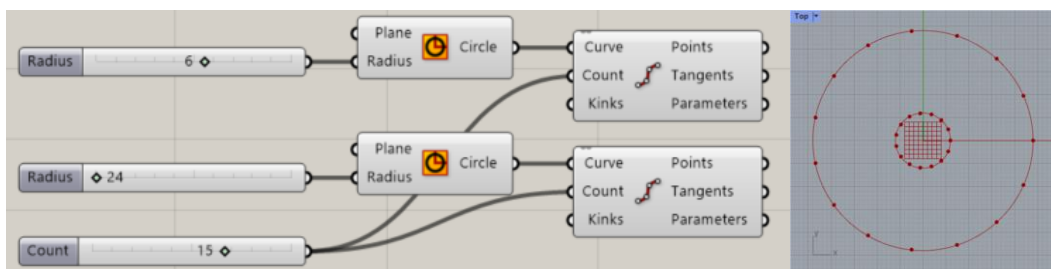

Step 2: Connect the output circle curve to the **Curve** input on the **Divide Curve** component. Repeat for the second circle.

Step 3: Double-click on the grasshopper canvas and create a slider by typing 5<15<20.

Step 4: Connect the output of the slider to the **Count** of both **Divide** components.

Step 5: Now drag the slider and watch the points increase and decrease.

Step 6: On the Grasshopper **Curve** tab, from the **Primitive** panel, pick **Line** and place it on the canvas, to the right of the **Divide** component.

Step 7: Connect the output Points from the first divide component to the **Line** input **Start Point**.

Step 8: Connect the output Points from the second divide component to the **Line** input **End Point**.
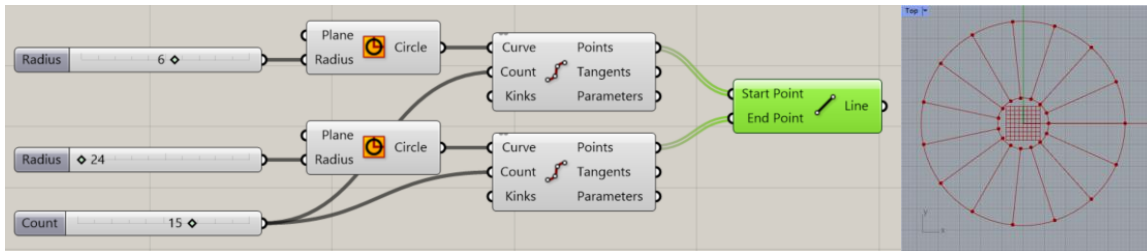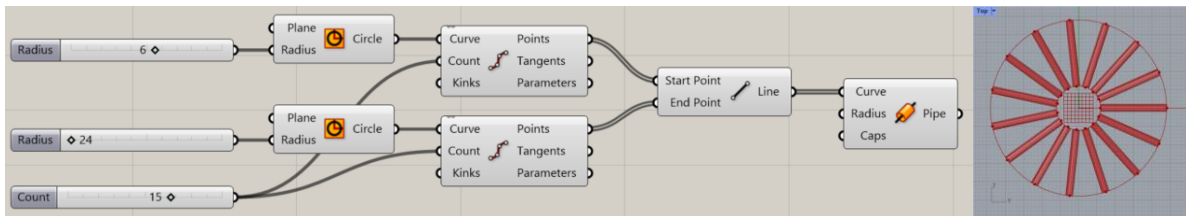

2.3. Pipe the Curves

The curves will be used to generate surfaces for the wheel and spokes of the wheel.

Step 1: On the Grasshopper **Surface** tab, under **Freedom** panel, click **Pipe** and place it to the Grasshopper canvas, to the right of the **Line** component.

Step 2: Connect the output from **Lines** to the **Curve** input of the first **Pipe** component.

Step 3: Double-click on the Grasshopper canvas and create a slider by typing 0.25<1<2.

Step 4: Connect the output from the **Number slider** to the **Radius** input of the first **Pipe** component.
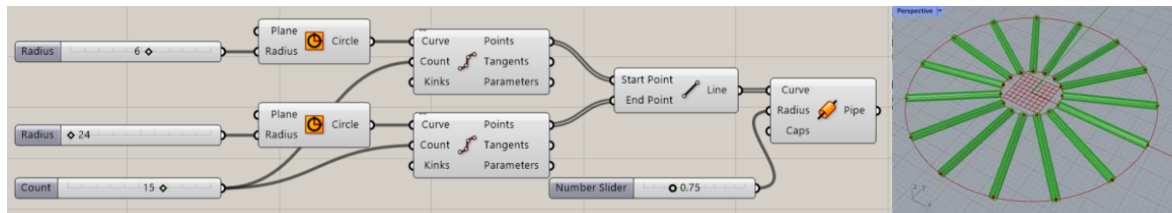
Step 5: Add a second **Pipe** component and place it on to the Grasshopper canvas, below the first **Pipe**.

Step 6: Connect the output from the first **Circle** curve to the input **Curve** on the second **Pipe** component.

Step 7: Add a third **Pipe** component and connect the output from the second **Circle** to the input **Curve** on the third **Pipe** component.
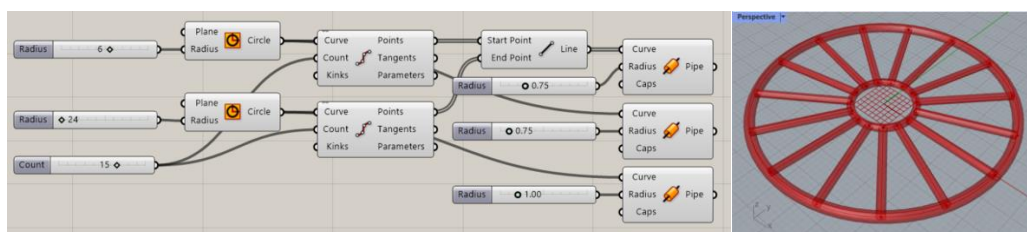
2.4. Orienting the wheel

Step 1: On the Grasshopper **Vector** tab, under **Plane** panel, click the **XZ plane** component and place two **XZ plane** on to the Grasshopper canvas, to the left of each circle component.

Step 2: Connect each **Plane** output from **XZ plane** to the Plane input of each **Circle** component. The entire wheel design is now oriented in the Front view or the World XZ plane.
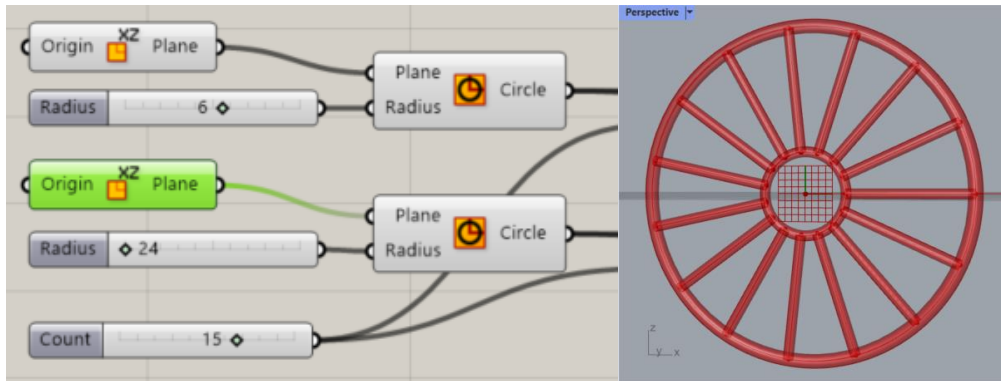


2.5. Preview the Wheel in Custom Color

The geometry is being previewed in the default Grasshopper colors. In this section, you will assign a colored material to the wheel and the spokes in Grasshopper.

Step 1: On the Grasshopper **Display** tab, under the **Preview** panel, drag and drop two **Custom Preview** components and place them to the right of each Pipe component.

Step 2: Connect the output from **Pipe** from **Pipe** components to the input **Geometry** of the **Custom Preview** components.

Step 3: On the Grasshopper **Params** tab, under **Input** panel, drag and drop a **Color Swatch** component to the Grasshopper canvas to the left of the **Custom Preview** components.
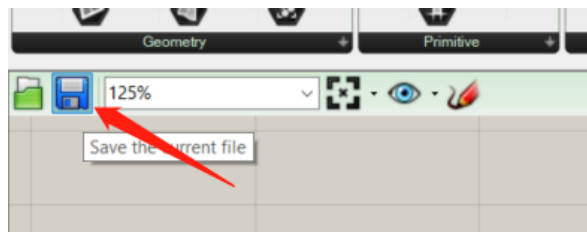
Step 4: Drag the output from the **Color Swatch** components to the input Material override on the **Custom Preview** components.
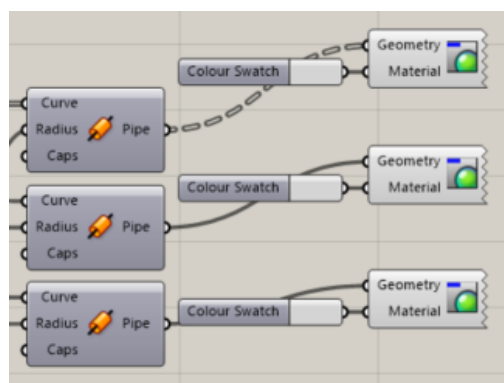
Step 5: Double-click on the **Color Swatch** components to select a custom color from the color picker dialog.
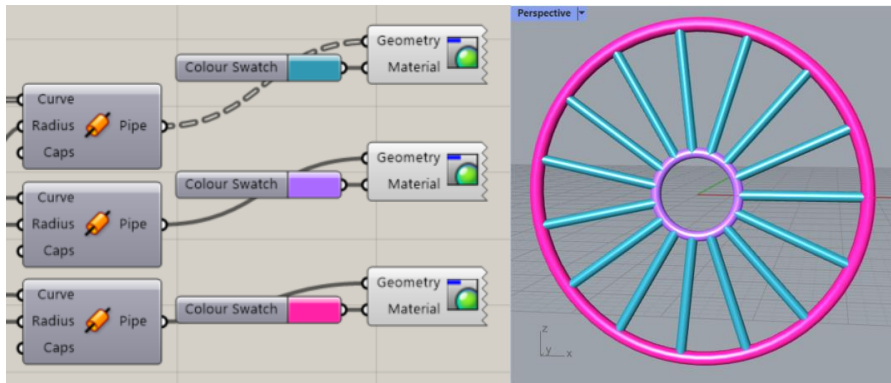
Step 6: Pick **Save Document** from the Grasshopper **File** menu, or pick the **Save** icon from the Grasshopper canvas toolbar.


2.6. Bake the Wheel

The wheel geometry is still only being previewed in Rhino. To send the geometry to Rhino for editing, rendering, printing and more, you will need to Bake from certain components. During the bake, there are options to assign the geometry to a target layer and group the geometry.

Step 1: On the Grasshopper **Params** tab, under **Primitive** panel. Click **Data** component. The **Data** component will make a copy of the input to be used collectively in another operation, like Bake.

Step 2: Connect the output from the three **Pipe** components to the input of **Data**. Hint: To allow multiple inputs, press the Shift key while connecting the outputs to the input (or use **Merge** component).
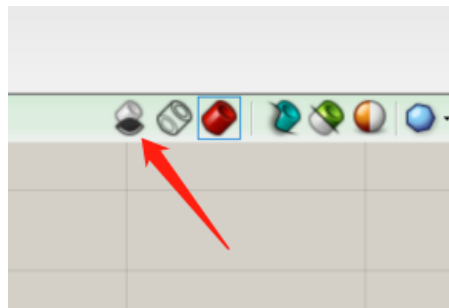
Step 3: Right click over Data and click Bake from the menu.
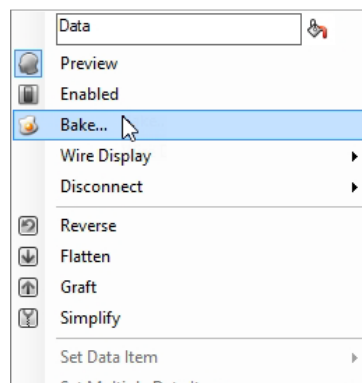
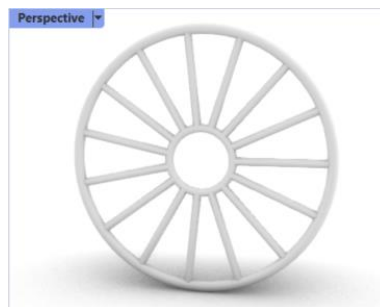Step 4: Select **Default** and **Yes, Please** to **Group** the output.
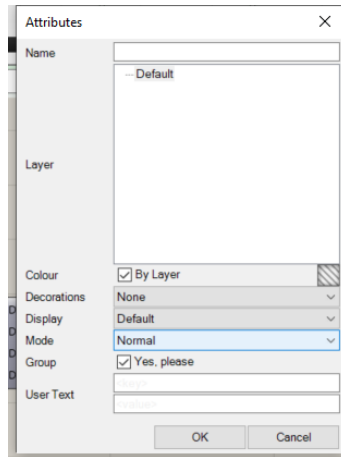
Step 5: In the upper right corner of the Grasshopper canvas, turn off the preview of the Grasshopper geometry.

Step 6: Minimize the Grasshopper window and you will see the model in Rhino.

Step 7: Render the wheel.

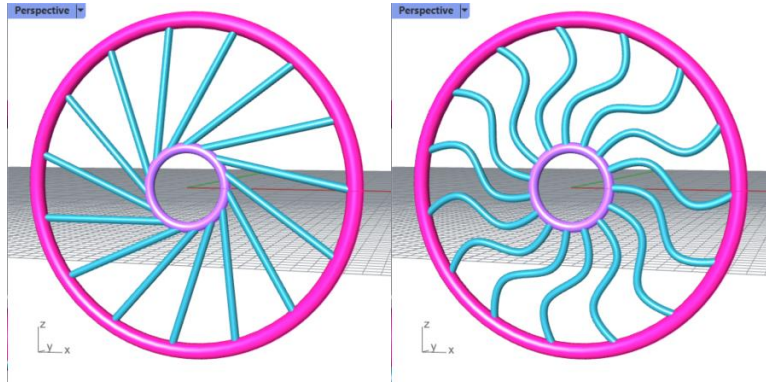# 3. Practice example

3.1. Deformed wheels
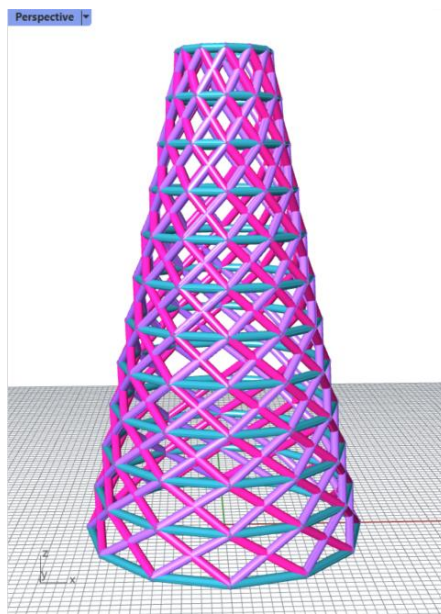
Fig. 4. Deformed wheels

3.2. Tower

Fig. 5. Tower