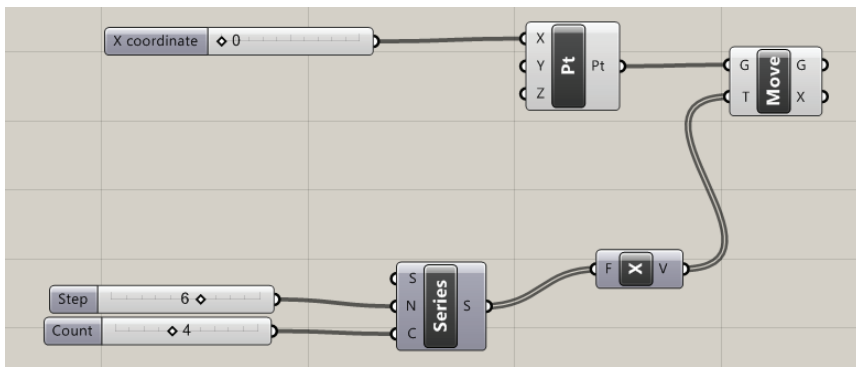# Rhino/Grasshopper_Tutorial 1

1.

Drag the 'line', the 'slider' and the 'move' components form the component panel. Edit the 'slider' to have integer numbers from 1 to 10. Connect the 'slider' to the x coordinate of the line component and connect the 'line component' to the G parameter of the 'move' component.
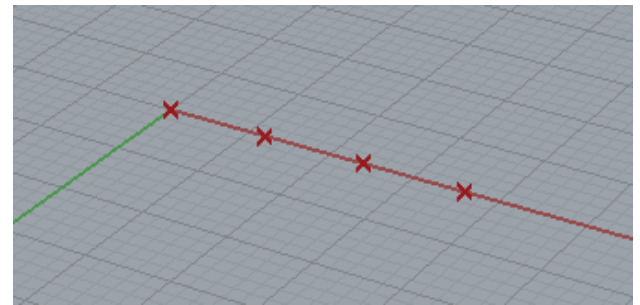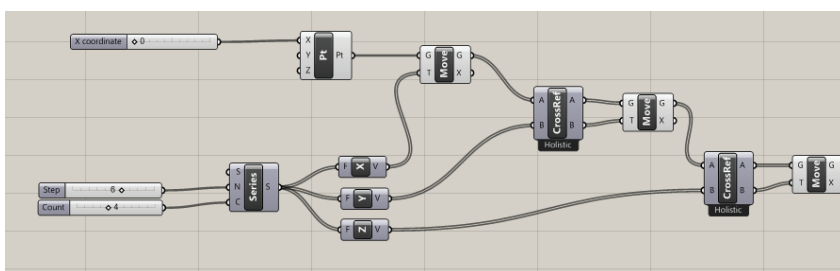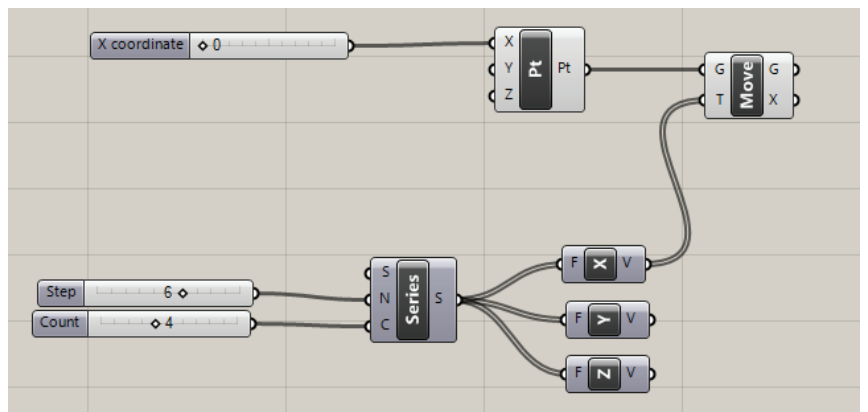

2.

Drag a series component and a 'unit vector (x)' and connect the 'series' component to the 'unit vector'. Take other two 'sliders' and connect one to the N parameter of the 'series' component and the other to the C parameter of the same component.

Arrange the first 'slider' to 6 and the second to 4. Now you will have a line of 4 points in the rhino viewport.
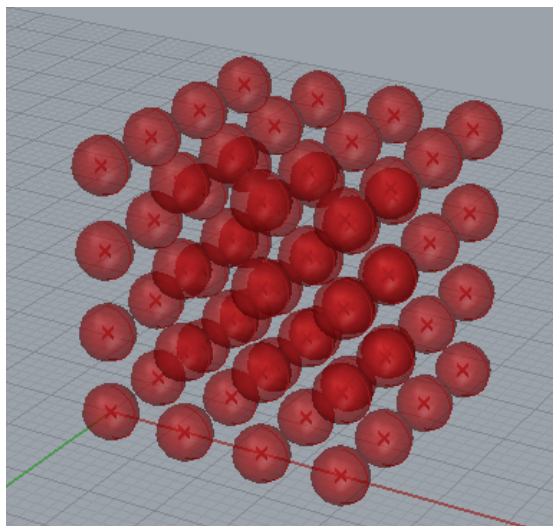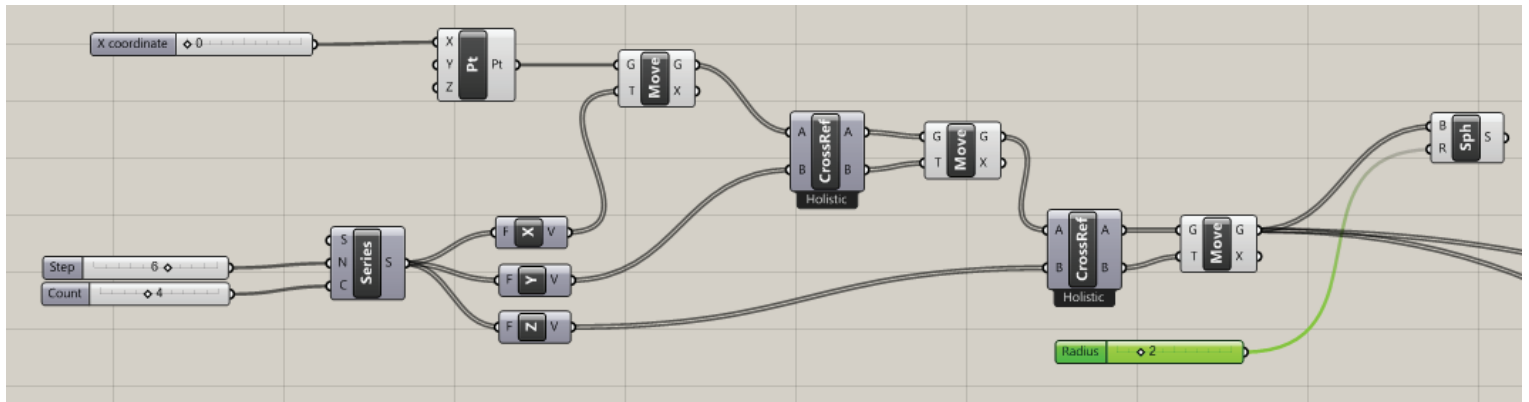

3.

Add a 'unit vector (y)' and 'unit vector (z)' to the 'series' component by holding Shift while dragging connection wires. In this way, the wire connecting the already 'unit vector (x)' will not disappear.

Insert two 'Cross reference' components and two other 'move' component. Connect them as shown in the image on the left. Then connect the 'unit vector (y)' and the 'unit vector (z)' to the B parameter of the components. Finally, connect the first 'move' component to the first 'Cross reference' components in order to link this step to the pervious one.

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4.
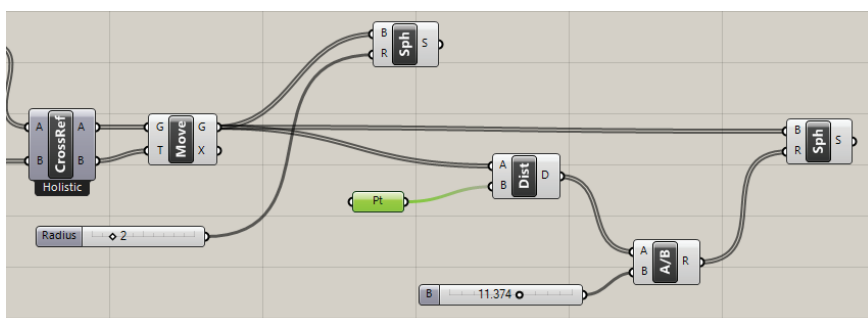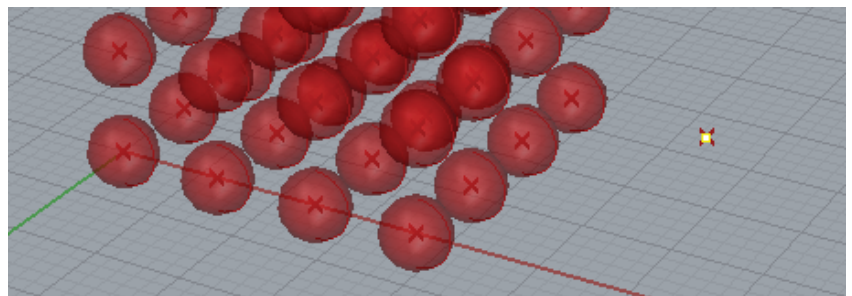

Add a 'sphere' component and another slider and connect the latter 'move' component to the 'sphere' component by wiring the outcome G of the 'move to the input B of the 'sphere'. Then connect the slider to the R input of the 'sphere'. You can also check if you are connecting the right component in the right input parameter by overlying the cursor of the mouse and then you will see which type of data is needed.

With this step you will see a cluster of spheres forming a cube as in the image in the left.

5.

In order to scale the spheres according to the position of an external point, you need to add first a point in the rhino viewport. Relate the point to a parameter in grasshopper by adding a point parameter and right clicking to 'set one point' and selecting the point in rhino.

Add a 'distance' component and connect the point parameter on the B input of the 'distance' component. Insert a 'divide' component and another slider. Connect the 'distance' to the A input of the 'divide' and the slider to the B input. Adjust the number in the slider until you get proper size for the spheres. Finally add a 'sphere' and connect it as shown in the left image. Now you can change the point in rhino and see how the size of the spheres change according to the distance.