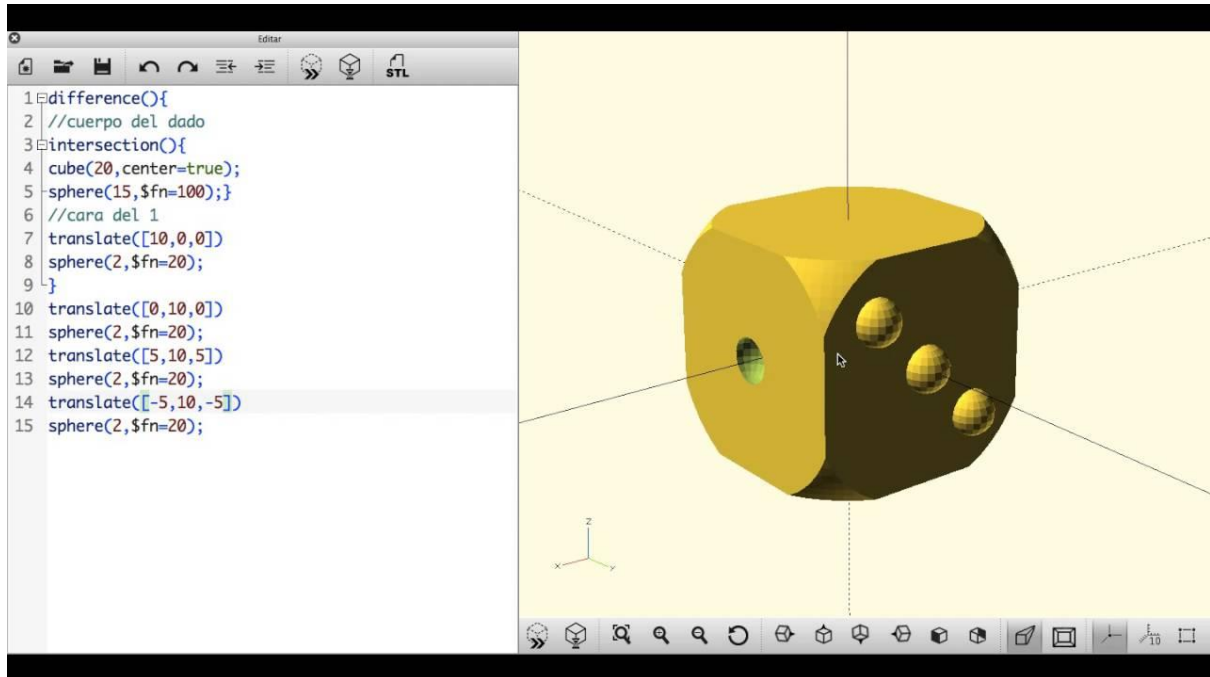# Dyson Day Outreach Activity Report: OpenSCAD and 3D Printing

The aim of this activity is to introduce the students to 3D modelling, 3D printing and a small amount of programming. During the activity, the students will be given a mini demonstration on how to use the free open source 3D modelling software 'OpenSCAD'. The students are split into pairs, for each pair: a laptop with the software and an example model keychain is provided. Each pair is given 20 minutes to customise the example keychain. At the conclusion of the activity, the design of each pair is taken and 2 copies are printed and returned at the end of the day.

OpenSCAD is a good introduction to 3D modelling. The software itself is very light, allowing it to run on any machine. OpenSCAD is a text based modelling software, the advantage of this is there is no complex user interface. The text based aspect can be intimidating to those unfamiliar with programming, however the language is relatively simple and the students are given an example with most of the commands they are reasonably expected to use.

The 10 minute demonstration at to start of the activity explains the basics of 3D modelling and the syntax of OpenSCAD. 3D Modelling in its most basic form, is creating primitive shapes, such as cubes or spheres, and combining these in different ways to create more complex models. In OpenSCAD the simplest primitives to work with are spheres, cubes and cylinders. These are generated using the commands 'sphere();', 'cube();' and 'cylinder();'. The properties of these are defined inside the brackets, for example: 'sphere(2,$fn=20);' generates a sphere in the centre of the workspace with a radius of 2mm and divided into 20 faces.

There are 3 ways to combine primitives: 'union(){}', 'difference(){}' and 'intersection(){}'. Union is effectively an OR operation, the final shape is the total of all shapes defined inside the curly brackets. Difference subtracts the second shape from the first shape inside the curly brackets. Intersection is effectively an AND operation, the final shape is only the volume which is occupied by both shapes, inside the curly brackets, at the same time.

These combination operations can be nested, i.e. take the difference of 2 unions. The order of these operations is important, especially when using the difference operation as this determines which shapes are included in the subtraction and which are not.

The final operations explained are simple transformations: 'translate(){}' and 'rotate(){}'. The x, y and z-axis are shown in the display window. Translate allows shift of the shapes inside the curly brackets by any amount in each axis. Rotate allows rotation about each axis, starting with the z-axis.

The example keychain model given to each student consists of the base shape, this is a union of shapes to make the keychain disk and some example shapes and text to stand out. These can be edited, for example the text and its position, size, font and style can be altered. Then the shapes to subtract from the base shape are defined, this includes to hole to attach to a keychain and some additional example text to be inlaid into the disk, as before this text can be edited. There are 2 highlighted areas for the students to insert their own simple shapes, for text, they are encouraged to copy the example. One area is in the base shape, by following the examples, they can add text or other shapes here. The other area is in the shapes to subtract, adding shapes here allows students to make holes or cut-outs in the keychain.

During the activity, multiple 4th year students were on hand to answer any questions. If the students had a problem, the techniques to finding a solution were explained, rather than just pointing them out. These were mainly syntax errors, especially from students forgetting to close brackets. By explaining why there is an error, the students were more likely to be able to find the exact mistake by themselves.

By the end each pair had created a unique design. 2 of each design was taken to the 3D printers, in total over 30 keychains were printed across 6 printers in 3 hours. Each student was able to take home their own custom keychain at the end of the day.

In conclusion, this activity was a success. The aims were achieved and the students had an opportunity to learn about practically designing something. By allowing students to design something for themselves and being able to take it home at the end of the day was a good motivator for applying themselves to the task. If this activity were to be repeated, additional time for the students to design their keychains would be useful. The demonstration could be more efficient, however significant time is needed to introduce the concepts in this activity as almost all students were unfamiliar. Additional time would allow the students to create more complex designs and explore the limitations of the CAD software, or would allow the students to come up with additional designs.

There were almost no risks involved in this activity. Each student was sat in a lecture theatre for 30 minutes with laptops with no charging cables, so no tripping hazards. There was no time to show the students the 3D printers, therefore there was no risk of burns of appendages trapped in machinery. No accidents occurred in this activity.