

# RoboDK Example

## Laser Cutting with Autodesk Fusion 360

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## Introduction

This section will help you create a basic project in RoboDK for robot simulation and offline programming. In this example a Motoman robot is simulated and programmed for a laser cutting application using Fusion 360 and RoboDK.

You can choose between two different methods: One using the feature extraction method and one using the laser cutting features from Fusion 360.

**Video:** Robot programming with Fusion 360 for a laser cutting application <https://youtu.be/MZcPXktyT8s>.

With the RoboDK plug-in for Fusion 360 you can easily load 3D models created in Fusion 360 to RoboDK. This plug-in allows you to program more than 50 different robot manufacturers and 500 robots directly from Autodesk Fusion 360.

The RoboDK plug-in for Fusion 360 is free if you have a RoboDK license.

**Tip:** Learn more about the RoboDK Add-in for Fusion 360 in the [RoboDK Add in for Fusion 360 section](#).

## Laser cutting example


To start the project, you will first have to select RoboDK's laser cutting example in the default library.



### Setup

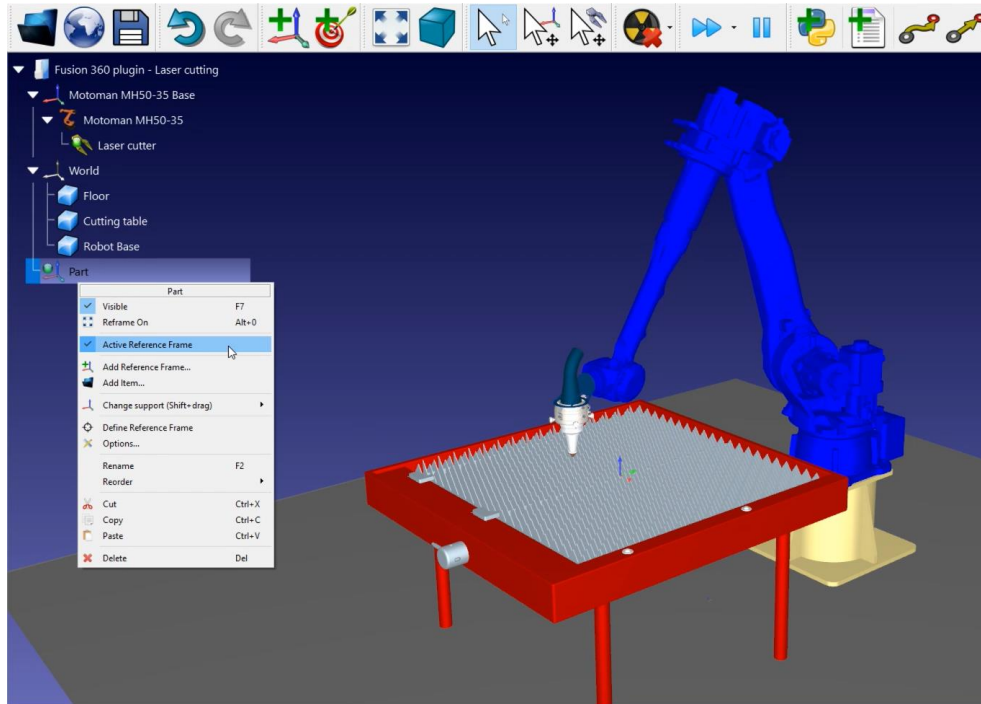
This example uses the Motoman MH50-35 robot, a laser cutting head, a stamped piece of metal and a laser cutting table.

**Tip:** [Follow this guide](#) to learn how to create a robot station step by step in RoboDK.

Load the station:

1. Select **File** →  **Open**
2. Locate the **Laser-Cutting** example from RoboDK's examples section:  
C:/RoboDK/Examples/**Plugin-Fusion-360-Laser-Cutting.rdk**.

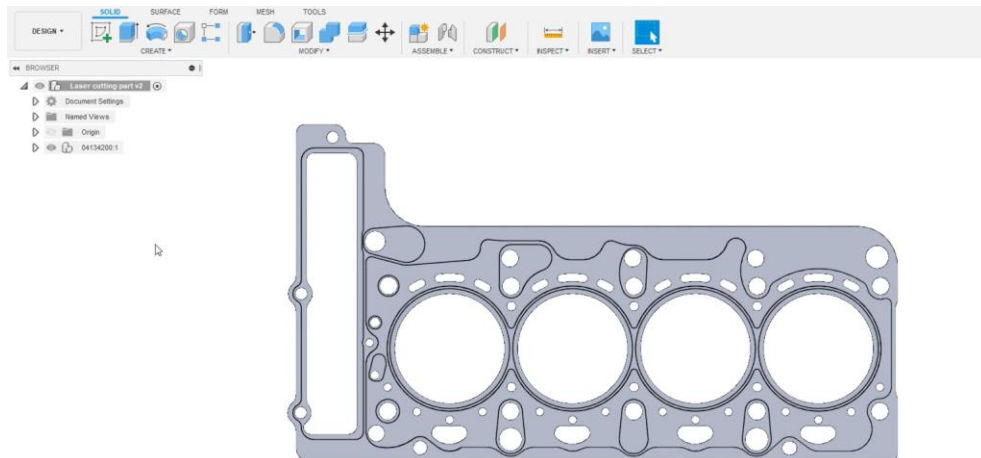
Note that the  reference frame called "Part" in the middle of the table is where the part will be imported. Make sure that this reference frame is activated by right clicking on  **Reference Frame** → **Active Reference Frame**.




Now that you have loaded the station you can open Fusion 360.

## Method 1: Feature extraction

Open your 3D object in Fusion 360. In this example the 3D object is a stamped piece of metal.

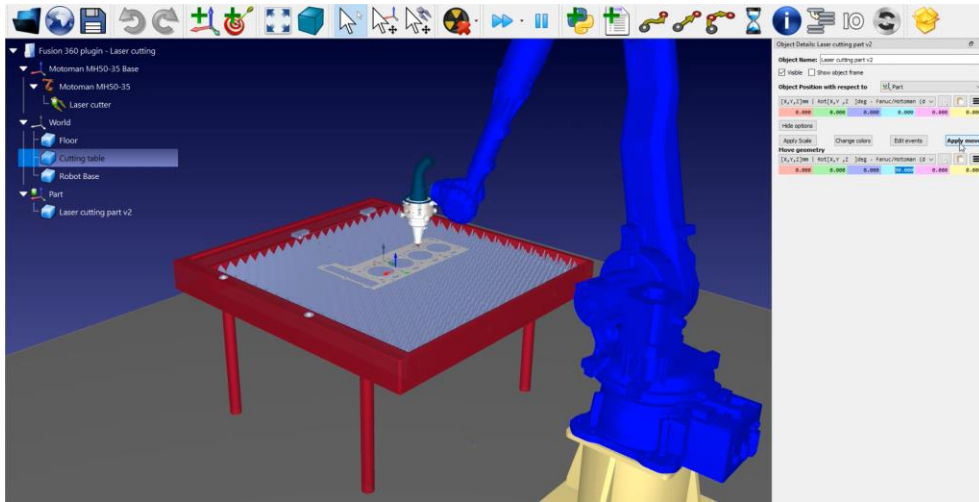


The goal of this project is to automatically create a robot program that will cut all those holes with a laser cutter. Programming that by hand would take too long.


1. Select the **Design** section of Fusion in the top left-hand corner.
2. Select the **Tools** tab. This is where you will find the RoboDK plugin.
3. Click on the button  to load the part to RoboDK.

The part has been imported in the active reference frame of your RoboDK station. Make sure that the part has been imported in the right orientation. In this example, you will have to modify the orientation of the part.

**Note:** The orientation of your part is defined by the orientation in which the part was designed in Fusion 360.

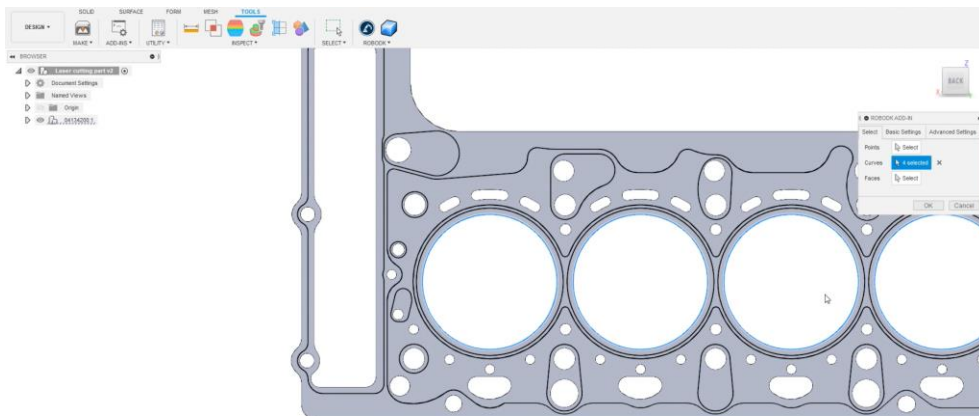


Follow the next steps to modify the orientation of your part:

1. Double click the  **Laser cutting part**. In this example the part is called Laser cutting part v2.
2. Click on **More options** in the open menu.
3. Go to **Move geometry** and enter 90 in the X axis (light blue) as shown in the image above.
4. Click on **Apply Move** to save the orientation.

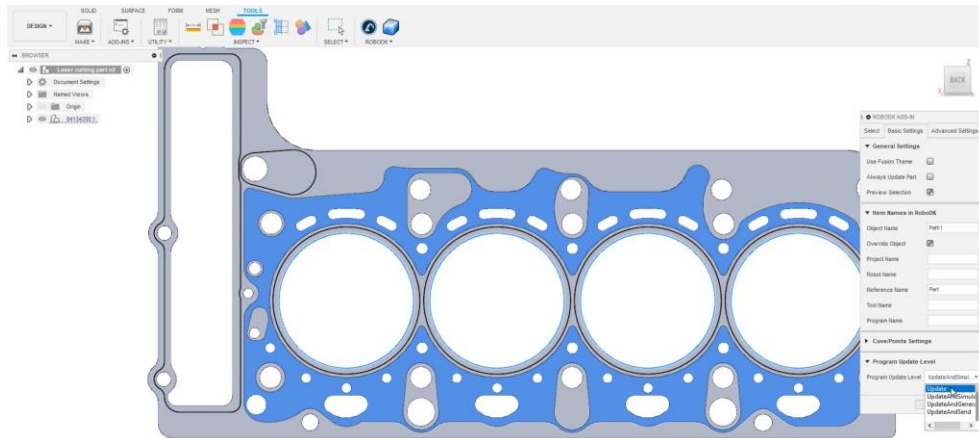
The RoboDK plugin has another feature in Fusion 360 that will help you import your cutting path to RoboDK.

1. Go back to Fusion 360.
2. In the RoboDK plugin menu, select **Load Curve(s)**.
3. Select **Curves** in the open menu.
4. Select the edges of the four big circles as shown in the image below.



Now you will need to give RoboDK an orientation for the robot tool.

1. Select **Faces** in the open menu.
2. Select the faces you wish the tool to be normal to as shown in the image below.





Now you can configure the settings:



1. In the same menu, go to **Basic Settings**.
2. Uncheck the **Use Fusion Theme** if you prefer to keep RoboDK in its default behavior.
3. Enter an **Object Name** to name your path. In this example it will be Path 1.
4. Enter a **Reference Name** you want it to be exported in. In this example it is named Part, the same as in the RoboDK station.
5. Select **Update** in the Program Update Level to have more options afterwards as shown in the image above. This is optional.
6. Click on **OK**.

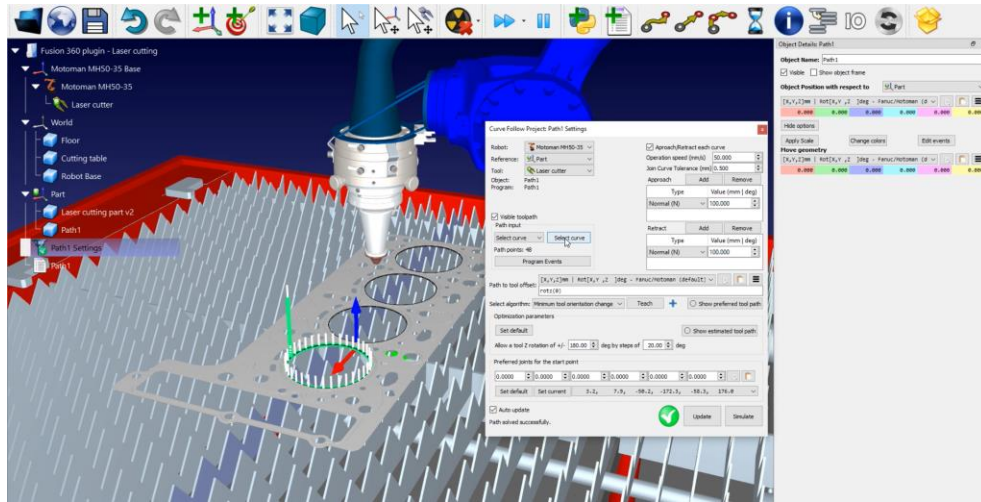
**Note:** In the **Advanced Settings** tab, you can find the **Linear Tolerance** that will control the number of points you will send to RoboDK. The smaller the number is, the higher the number of points and therefore the precision of the path.

You should now be able to see the imported path in RoboDK. In this example, the orientation is wrong again and you will need to modify the orientation as you did for the part before.

1. Double click your path, in this example it is  **Path 1**.
2. Click on **More options** in the open menu.
3. Go to **Move geometry** and enter 90 in the X axis (light blue).
4. Click on **Apply Move** to save the orientation.

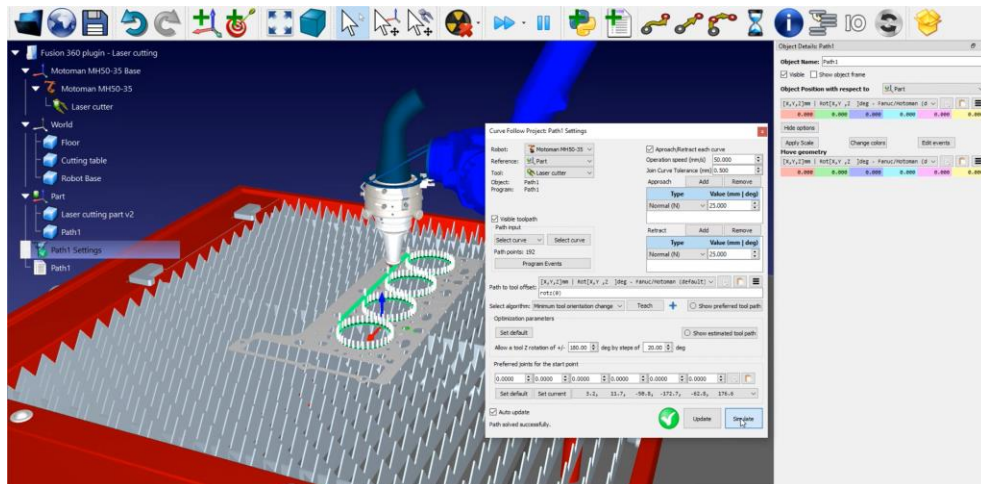
The next step is to verify if the order in which the path sections will be executed is correct by selecting  **Path1 Settings** → **Update** → **Simulate**. In this example, only one of the four circles has been selected as shown in the image below.

1. Double click on your path settings, here  **Path1 Settings**.
2. Make sure that the robot, the reference, and the tool are the right ones.
3. Click on **Select curve**.
4. Select the other circles.
5. Click  **Done**.



**Note:** You can switch the direction of the path by right clicking on it and selecting **Switch sense**.  
 You can now notice that the approach and retract movements are a bit too long in terms of length.  
 You can easily modify those values by:

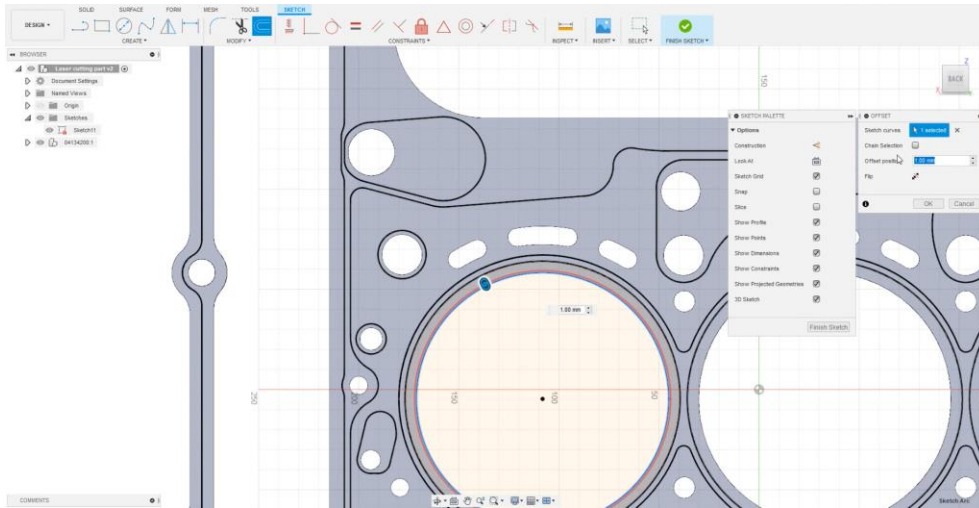
1. Double click on your path settings, here  **Path1 Settings**.
2. Enter a smaller value in the **Approach** and **Retract** boxes. In this example, you can enter 25mm.
3. Click on **Update**→**Simulate**.



The next step is to consider the radius of the laser. In this example, the laser is cutting too much material.  
 How to set up your laser radius:

1. Go back to Fusion 360.
2. Got to the tab **Solid**→**Create Sketch** to create your sketch.
3. Select one circle and click on **Offset** in the top menu to create a new path.
4. Considering that the diameter of the laser beam is 2mm, the radius will be 1mm. Enter 1 in the **Offset position** box as shown in the image below.
5. Repeat those steps for the 3 other circles.
6. Click **Ok**.







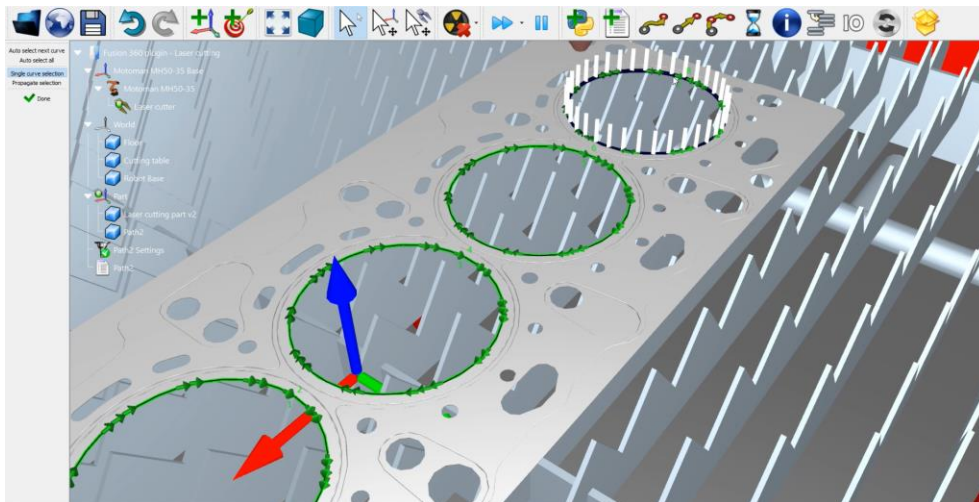
You can now load the curves into RoboDK.

1. Go to the **Tools** tab.
2. Click on **RoboDK→Load Curve(s)**.
3. Select **Curves** and click on the sketch in each circle.
4. Select **Faces** and click on the face you wish the tool to be normal.
5. In this example, the **Object Name** (Basic Settings) is Path2.
6. Click **Ok** to import the path into RoboDK.

You can modify the orientation of Path2 as explained above. If you come closer, you can see the difference between the two paths. You can now delete the Path1.

The next step is to verify if the path sections you loaded are correct by:

1. Selecting  **Path2 Settings→ Update** to make sure that the path is at the right position.
2. Click on **Select curve** and select the other circles as shown in the image below.
3. Select  **Done**.



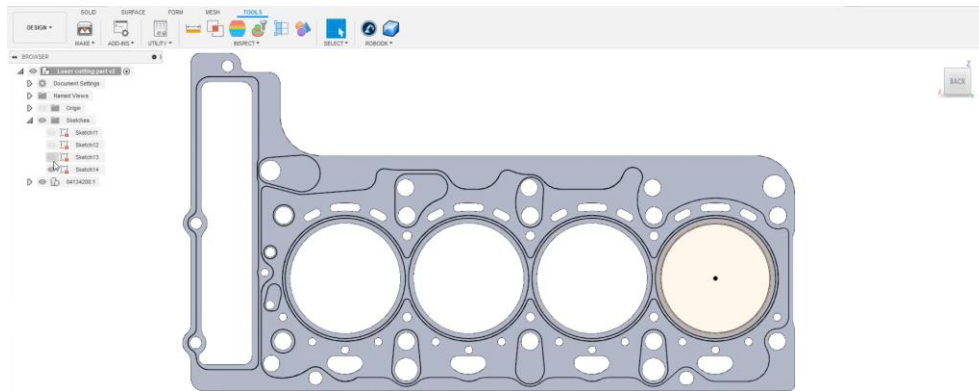
You can reduce the size on the **Approach** and **Retract** movements as explained above. In this example, you can enter again 25mm. Don't forget to **Update→Simulate** your Curve Follow Project to save your settings. You could do the same steps for all other holes of this part.

This way of building a laser cutting program works for RoboDK's Fusion 360 plugin, but it also works for all the CAD plugins.



## Method 2: Fusion 360 laser cutting features

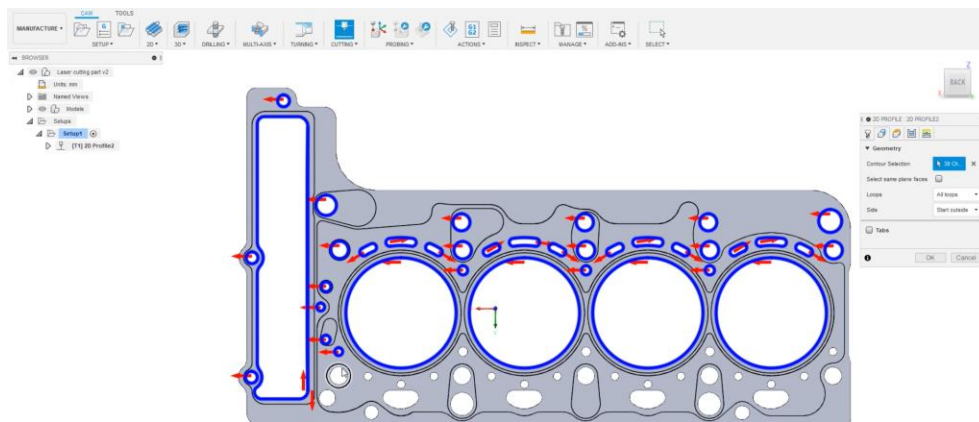
To use the Fusion 360 laser cutting features, go back to Fusion and hide the sketches you did before as shown in the image below.



Switch from the Design section in the top left-hand menu to **Manufacture**. In the tool bar click on the 2D cutting feature.

1. Select a **Tool**. In this example it is a 1 mm wide cutter.
2. Go to the **Geometry** tab in the same menu and **select the circles** you want to cut. In this example, all holes of the stamped piece of metal are selected as shown in the image below.
3. Click **OK**.

**Note:** Make sure that the cut is on the right side of the path.





You can simulate the path in Fusion 360 by right clicking on your path and selecting **Simulate**. In the Setup, make sure that you are using the same origin as the model. This way, it will be exported at the right position.

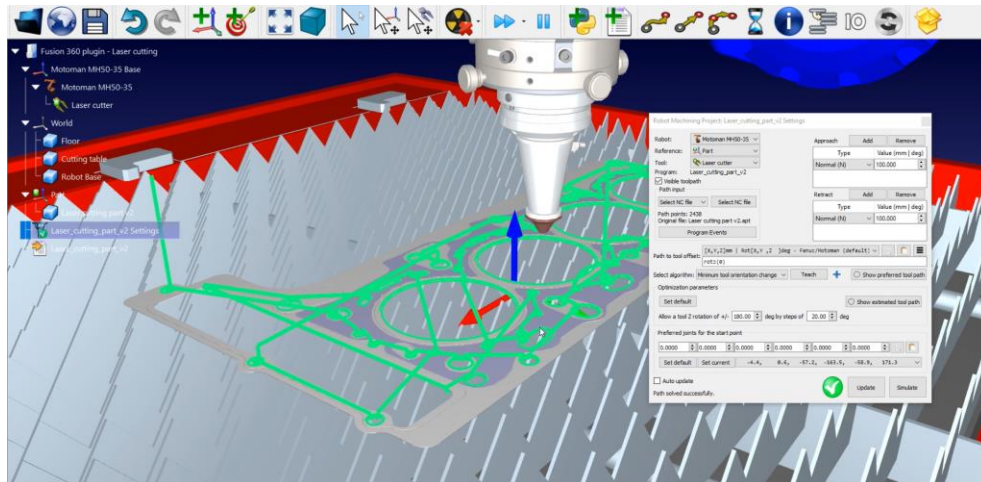
1. Right click on **Setup1**→**Edit**.
2. Make sure that **Model origin** is selected in the Origin box.
3. Select **Ok**.

Now you can transfer your sketch to RoboDK by selecting **Tools**→**Load CAM project in RoboDK** in the Fusion 360 tab.

You should see the imported sketch in RoboDK. Make sure to delete the previous path, in this example Path2.


Double click on  **Laser cutting part v2 Settings** to open Robot Machining Project. You can already see the green path on top of the part. Click on **Update**→**Simulate** to verify this path.

**Note:** You can hide the normals to better see the path by right clicking on  **Laser cutting part v2 Settings**→**Visible normals**



The only thing to manage is the **Program Events**:

1. Click on **Program Events**.
2. Let the first three text boxes free as you are not doing a machining project with a spindle, multiple tools, or some specific RPM to reach.
3. Check the **Path start** box and write **LaserOn** to activate the laser only when the path starts
4. Check the **Path finish** box and write **LaserOff** to deactivate when the path ends.
5. Click **Ok** and **Update**.

To generate your robot program right click on  **Laser cutting part v2** → **Generate robot program** or press **F6**. A new window will appear with the program post processed for the Motoman controller.

The .pg file is now ready to be transferred to your robot controller.