

Adding Boundary Conditions:

For 2D domain, we can add Boundary Conditions at points and lines, for a 3D domain we can add Boundary Conditions at points, lines (curves), and surfaces.

In toPy we have three types of Boundary Conditions:

- Displacement;
- Force;
- Fixed Elements (elements that we determine for not be excluded during the procedure of the Topology Optimization).

In order to add the Boundary Conditions we need to create Physical Groups using the Gmsh native functions, the names that are given to these Physical Groups are the Boundary Conditions.

To add a Boundary Condition of Displacement:

The name of the Physical Group should be, ***displacement\_x\_a\_y\_b\_z\_c***, where ***a***, ***b*** and ***c*** are the magnitude of the correspondent for each direction. If you want to add a Fixed Support, you just need to input a, b and c equal to zero. If you need to set free the directions the value of a, b or c need to be ***nan***.

To add a Boundary Condition of Force:

The name of the Physical Group should be, ***force\_x\_a\_y\_b\_z\_c***, where ***a***, ***b*** and ***c*** are the magnitude of the correspondent for each direction.

To add a Boundary Condition of Fixed Elements:

The name of the Physical Group should be, ***fixel\_xx***, where ***xx***, is an alphanumeric character of your choice just for a better organization, and in Gmsh we can not use the same name for different Physical Group.

If you need to add a Boundary Condition at a specific point, you can use the function *Boundary at Node* in the side menu. To use the function you need to generate the mesh first, then you can search for the tag of the node at the nearest desired point.

Just a remainder, the toPy is able to perform the Topology Optimization only on triangular elements for 2D domain, and tetrahedral elements for 3D domain.