

# TUTORIALS

---

**Tutorial 1** : How to use the tool

**Tutorial 2** : How to modify the model

## Tutorial I: How to use the tool

1. Enter the  $\Delta V$  corresponding to your mission.
2. Enter the blackbox corresponding to your payload and subsystems (i.e. everything which is not tanks, fuel, or structure).
3. Enter the g-force corresponding to the launcher used to launch your lunar lander into space.
4. Enter the g-force corresponding to the landing on the moon.
5. The tool gives you the results : structure mass, fuel mass, tanks' size...
6. To see the lander : open the file « Visualization » with gmsh.
7. To run the optimization : launch nastran95 with the file « nastran\_input.inp »
8. Now you can open the file nastran\_input.out in the tool's folder and have a look to all the outputs : mass, stresses, moments of inertia...

## Tutorial 2: How to modify the model

1. Open GMSH and create a whole new model or modify the `gmsh_input.geo` file.
2. Make sure to have physical groups and to write the same meshing commands which are at the end of the `.geo` file.
3. Check if you have no errors and a coherent mesh when your model is opened with GMSH.
4. Save the mesh file with GMSH.
5. Change the name and the extension of the file so it is named : `Mesh.inp`.

GMSH only provides CBARs when meshing. Hence you need to do some modifications in the `.inp` file if you want CRODs or CBEAMs. Also, you have to write the orientation vector for CBARs.

These modifications can be done manually but if the model have a lot of elements it can be quite long. This is why a corrector file is provided. This corrector file was used for the lander already used in the tool. It is a template that you have to modify so it corrects what you want.

4. Use your corrector file.
5. Duplicate the new mesh file.

6. Write, at the beginning of the duplicated file, the whole case control deck (cf. nastran95) + elements' properties + material properties + AUTOSPC,1

The AUTOSPC function will constrain singularities automatically. However this function is not compatible with the optimization function POPT. Therefore, you have to follow the next steps if you want to optimize your structure.

7. Launch the duplicated file with nastran.
8. Look at the constraints added and write them manually in your .inp file. To have an example, open the mesh file already in the tool and look at grid point 1.
9. Delete the AUTOSPC,1 line and write the POPT command like on the existing file.

Now, you should have an analysis that runs and optimizes the structure.

You have the possibility to implement this analysis in the python code in order to have the different inputs — payload, g-force, fuel mass — modified automatically after the mass estimation. Allowing you to simply launch nastran to get the optimized structure related to your mass estimation.

Follow the next steps to do this.

10. Write everything you add to the duplicated file in the python code. Your new lines must be written from line 176 of the python code, the same way it was written on the original code. (Do not forget to use « %s » for the inputs in order to connect them to the mass estimation outputs and to the inputs you enter in the python code)

11. Now you have a new tool !