

#####

WARNING :

This method is in no way obligatory! if you don't want this option and you want the normal printer.cfg you have to take the one that is not in the With Bed Mesh Area folder and without the bed_mesh_area.cfg file

NOM	MODIFIER
With Bed Mesh Area	11-05-24
bed_mesh_area.cfg	11-05-24
printer.cfg	11-05-24

#####

Mes chers camarades bien le bonjour !

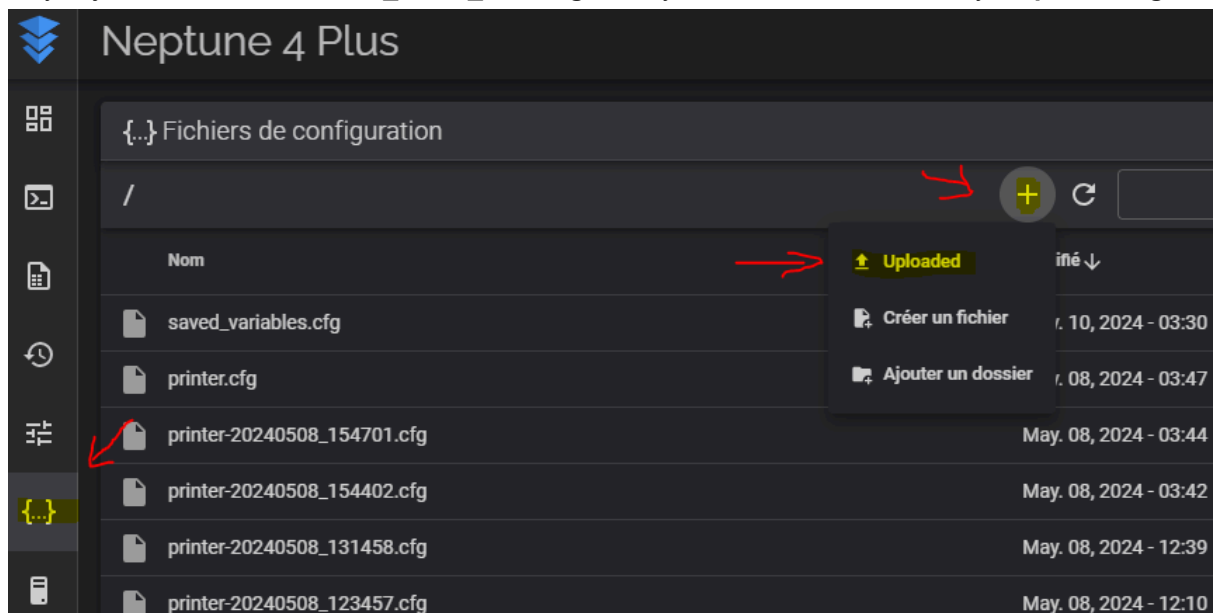
Here I will explain to you how to take advantage of the Bed Mesh Area option that comes from this GitHub:

<https://gist.github.com/ChipCE/95fdbd3c2f3a064397f9610f915f7d02>

So you have several solutions; the first you want to test but don't want to change the entire printer.cfg

The first one :

So you just need to add the **bed_mesh_area.cfg** file in your fluid interface next to your **printer.cfg**



So you go to the **Configuration** tab, you press the little **+** and finally **Uploaded**, there you will select the **bed_mesh_area.cfg** file that you downloaded beforehand!

You have one step left: add the line:

```
[include bed_mesh_area.cfg]
```

```
✕ printer.cfg
60 #
61 #####
62 # Base Config
63 #####
64
65 View 'include' documentation
65 [include plr.cfg]
66 View 'include' documentation
66 [include bed_mesh_area.cfg]
67 View 'mcu' documentation
67 [mcu]
```

Once the line has been added you must modify your start gcode with the one that I have always made available to you on my github and removed it; in front of the long line:

```
Gcode START and END for Orca.txt Gcode START and END for cura.txt Gcode START and END for Prusa-Orca.txt Gcode START and END for cura.txt Gcode START and END for Prusa-Orca.txt Gcode
1 //!\!\!\!\ Si vous voulez activé le bed mesh area il faut retirer le ; devant la ligne 12 //!\!\!\!\!\!\!\!\
2 //!\!\!\!\!\!\!\!\ If you want to activate the bed mesh area you must remove the ; in front of line 12 //!\!\!\!\!\!\!\!\
3
4 START GCODE
5
6 ;ELEGOO NEPTUNE 4/4Pro
7 M220 S100 ;Set the feed speed to 100%
8 M221 S100 ;Set the flow rate to 100%
9 M140 s[first_layer_bed_temperature]
10 G90
11 G28 ;home
12 ;BED_MESH_CALIBRATE AREA_START=(first_layer_print_min[0]),(first_layer_print_min[1]) AREA_END=(first_layer_print_max[0]),(first_layer_print_max[1])
13 M104 s[first_layer_temperature]
14 G92 E0 ;Reset Extruder
```

You can now enjoy the bed mesh area!!

The second method you replace your printer.cfg with the one which is in the With Bed Mesh Area folder and of course as for method one you replace your start gcode.

Crédit :

Julien Mairy / Printer'n Beer / SmartHome42

Youtube : <https://www.youtube.com/@printernbeer>

Page facebook : <https://www.facebook.com/Smarthome42>

Groupe facebook où je suis actif :

<https://www.facebook.com/groups/impressions3dfr>

<https://www.facebook.com/groups/elegooneptunefr>

Github : <https://github.com/mairyj/Elegoo-Neptune-4-Series>

Instagram : <https://www.instagram.com/mairyjulien/>

TikTok : <https://www.tiktok.com/@julienmairy>

Cults3D : <https://cults3d.com/fr/utilisateurs/mairyjulien/>

Printables : https://www.printables.com/@JulienMairy_174899

Makerworld : <https://makerworld.com/en/@mairyjulien>