Figures Tutoriel Chapter X

Representation of the drainage network in urban and periurban areas using a 2D polygonal mesh composed of pseudo-convex elements

The various steps presented in this document are part of a series of exercices the objective of which is to reproduce most of the figures included in this chapter. Two steps can be distinguished: the first one that only uses the TriangleQGIS plugin for the triangulation of bad-shaped elements, and the second one that also allows the second step, called dissolution and that uses the Geo-PUMMA toolbox, based on Python and Grass scripts and functions.

Part A: (QGIS-Windows)

Installation of the TriangleQGIS plugin in Windows

Getting the following figures included in the chapter under numbers:

x2;x28;x30;x31;x34;x35;x36

Part B: (QGIS-GRASS-Virtual box)

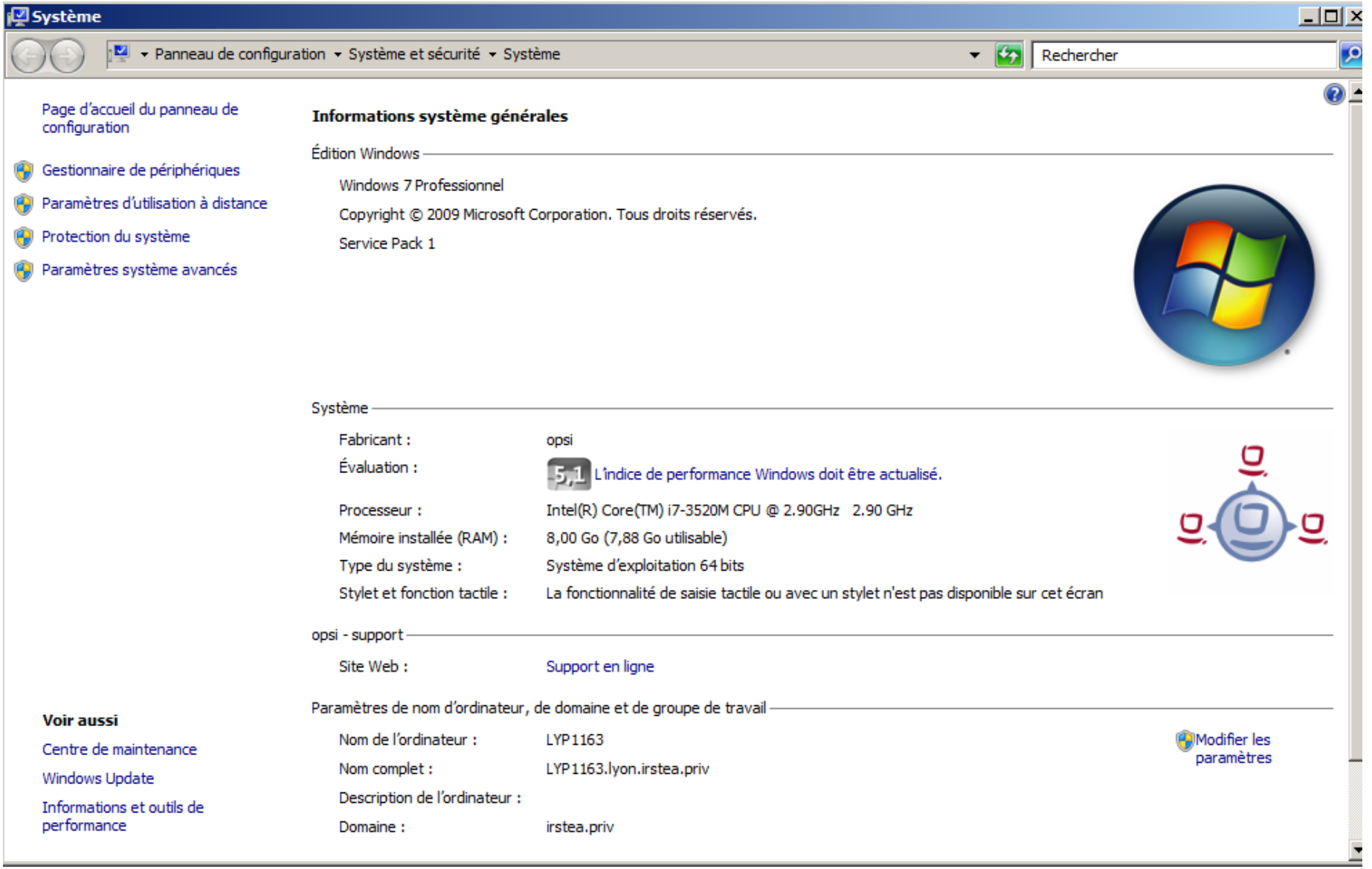
Dissolution of the triangulation and application to the Mercier catchment

Getting the following figures included in the chapter under numbers:

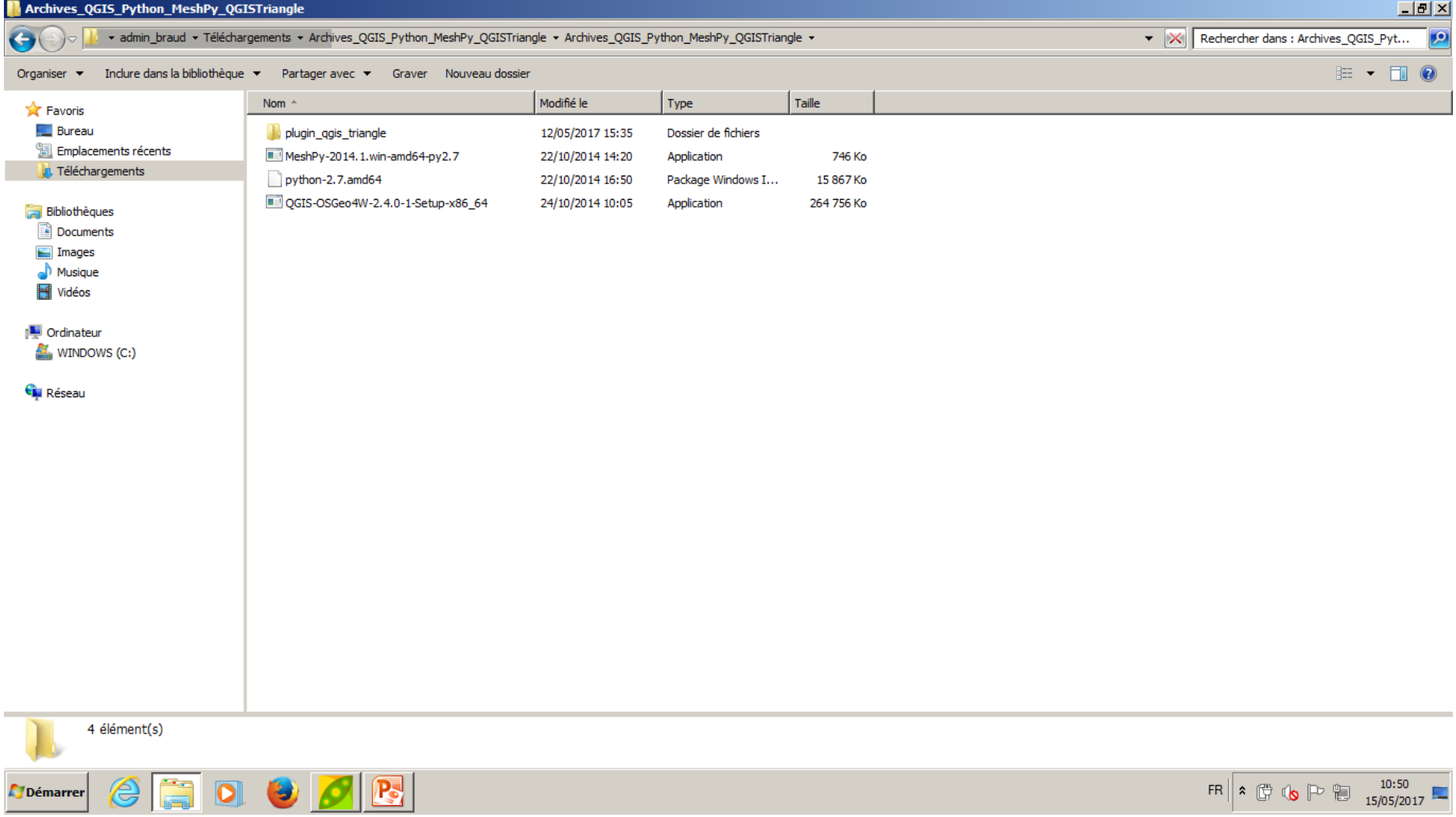
X34;x35;x36;x37

Part A: (QGIS-Windows)

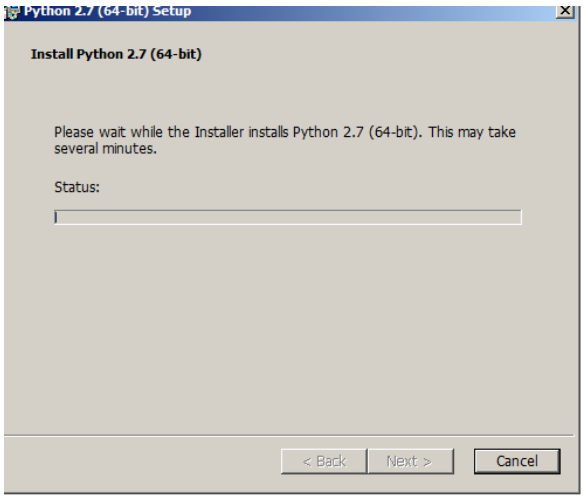
Installation of the TriangleQGIS plugin in Windows

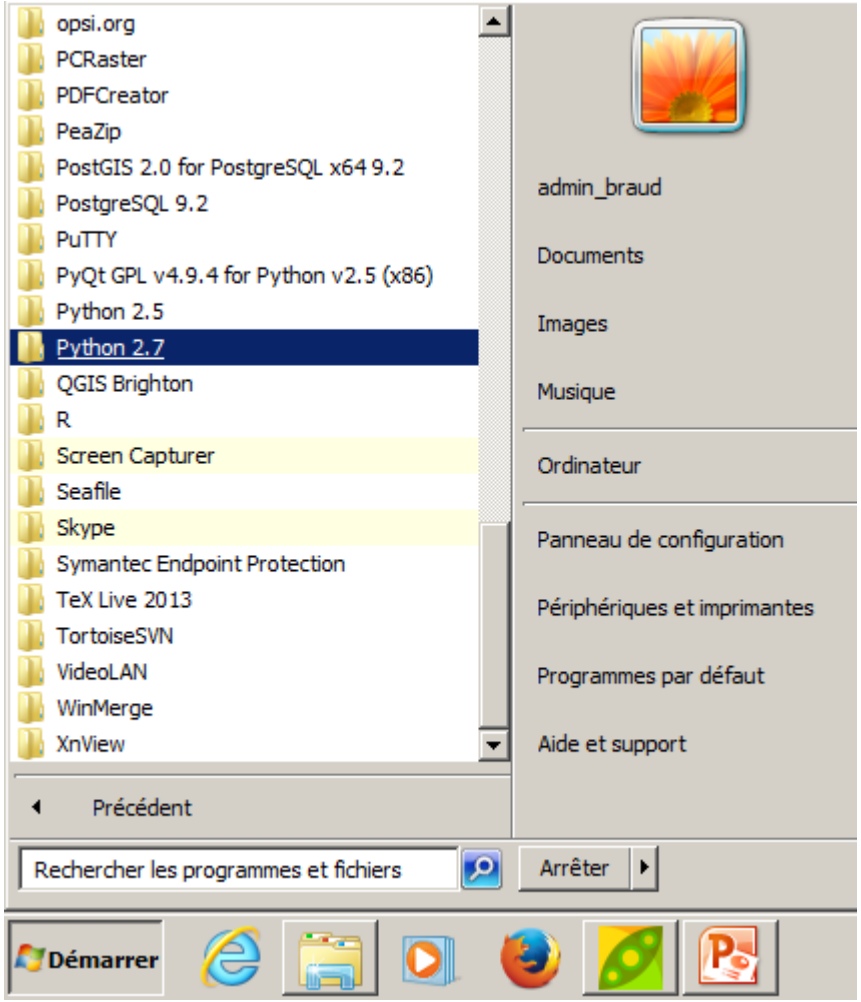


Presentation of the general characteristics of the computer



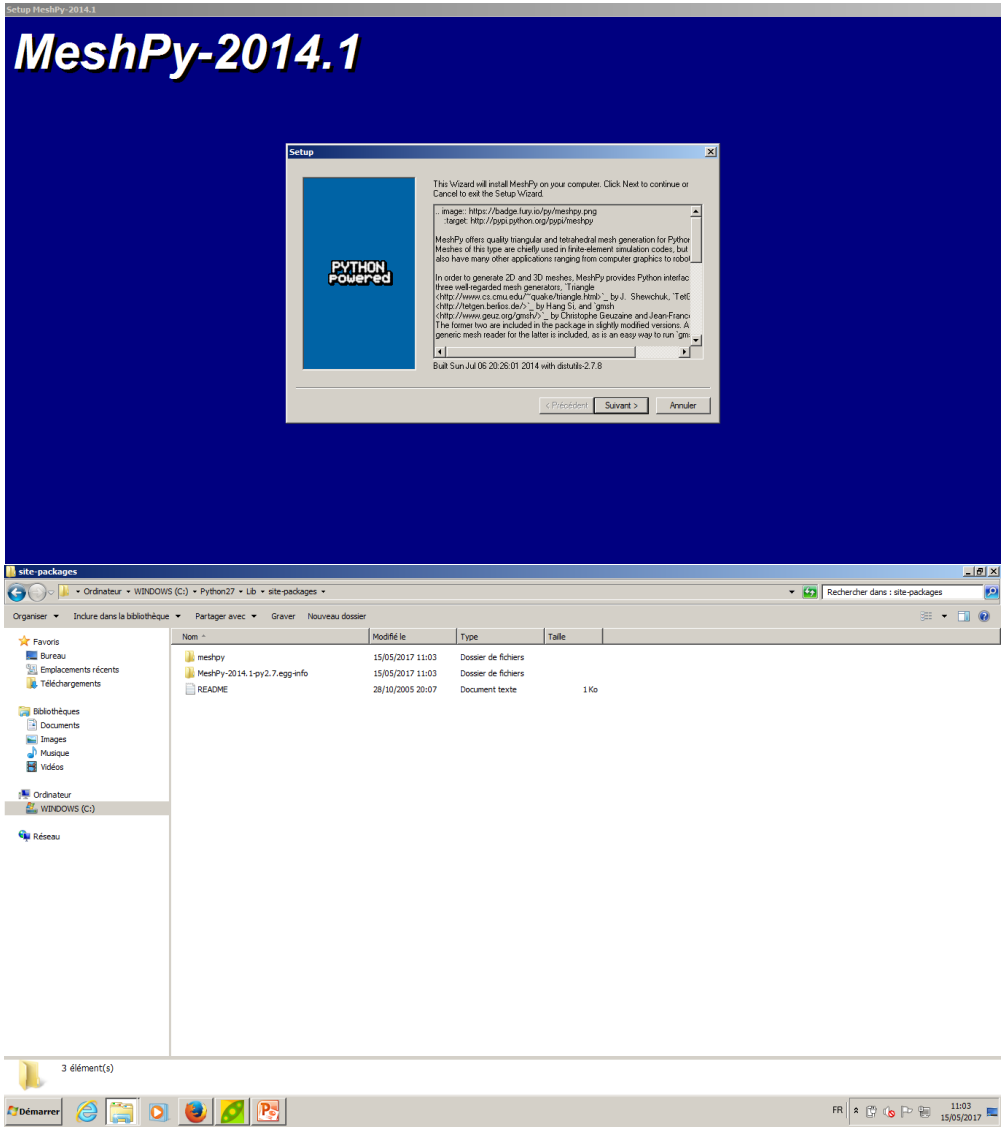
Folder containing the installation archive of QGIS, Python, MeshPy et QGISTriangle:
Install QGIS 2.4 or higher
Install Python 2.7
Install MeshPy





Step: Installation of Python 2.7

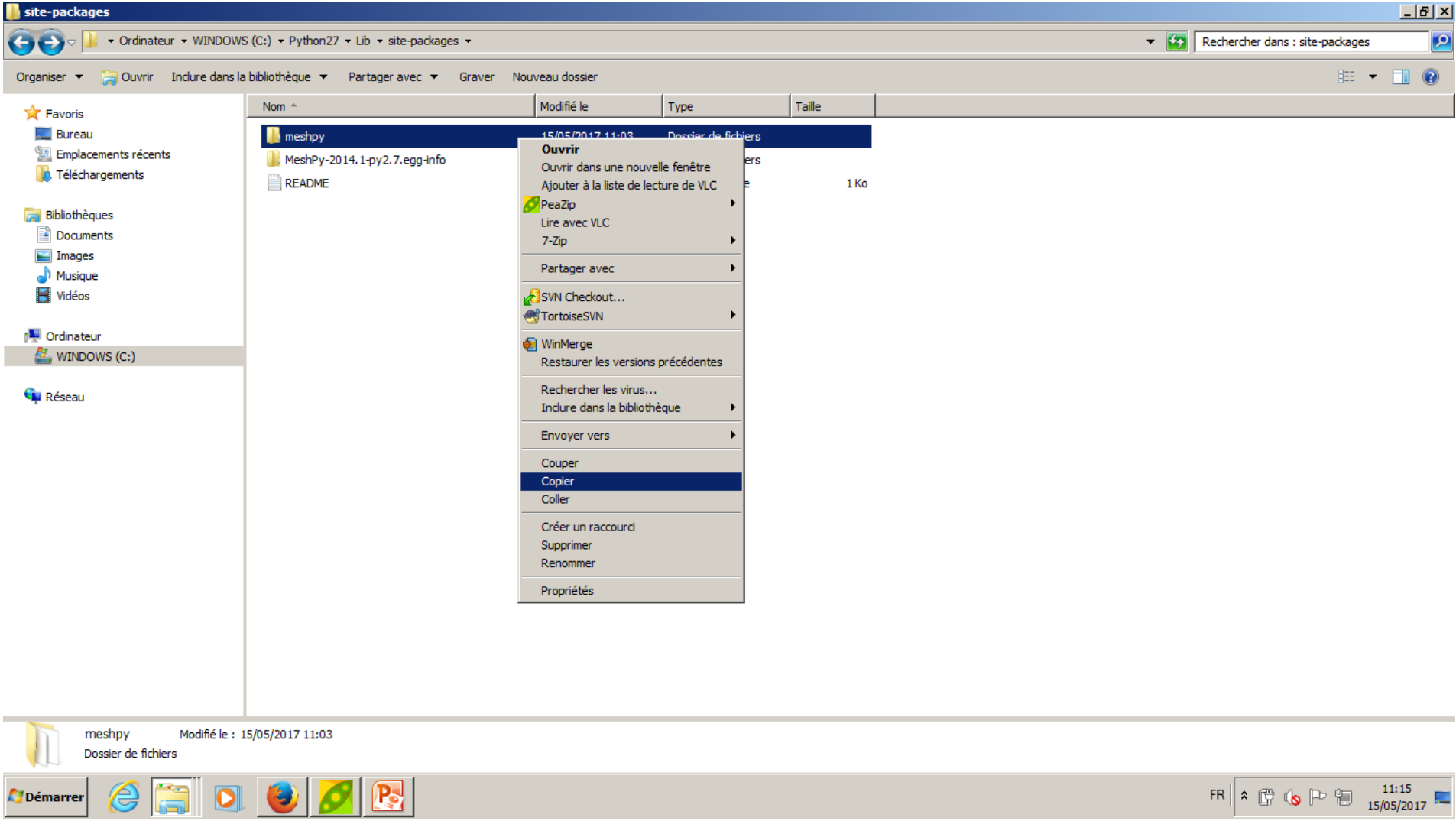
Verify that Python is correctly installed in the starting bar menu



Step: Installation of MeshPy

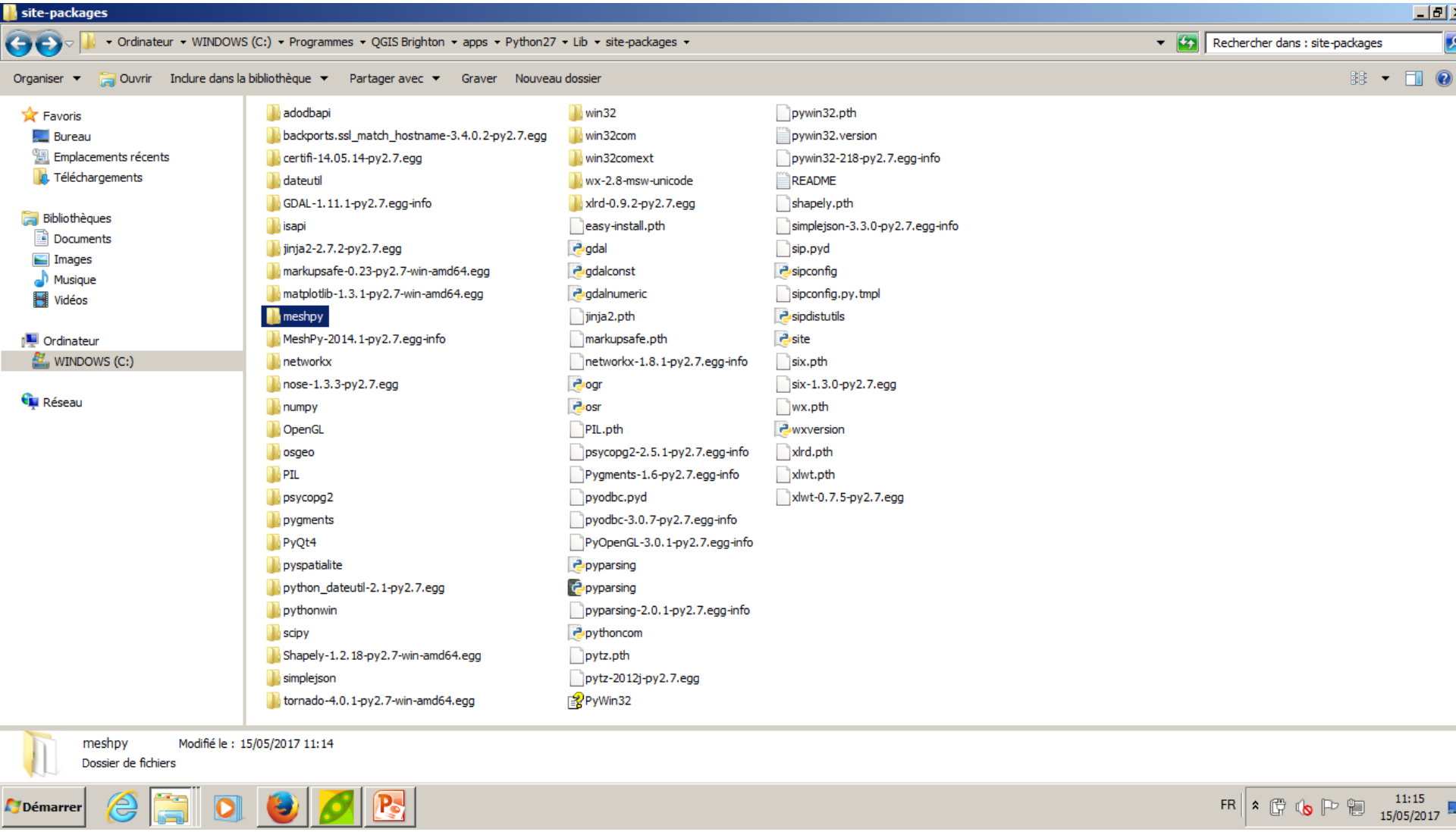
If everything is OK, the MeshPy library should be in the folder:

C:\Python27\Lib\site-packages



Step: Installation of MeshPy in QGIS

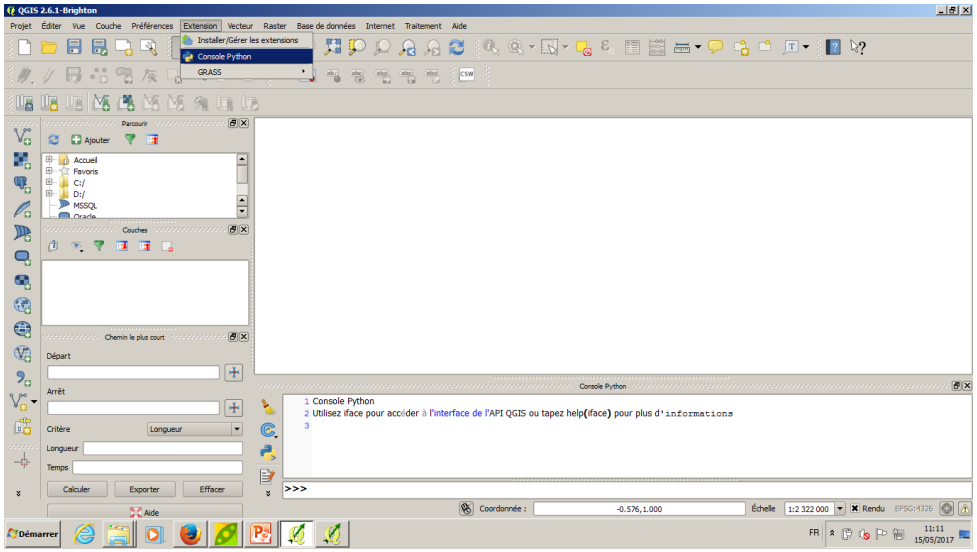
Copy the MeshPy folder from the folder: C:\Python27\Lib\site-packages



Step: Installation of MeshPy in QGIS

Copy the MeshPy folder in the folder:

C:\Program Files\QGIS Brighton\apps\Python27\Lib\site-packages



Step: Verify that the library is well installed in the Python from QGIS:

-Open QGIS

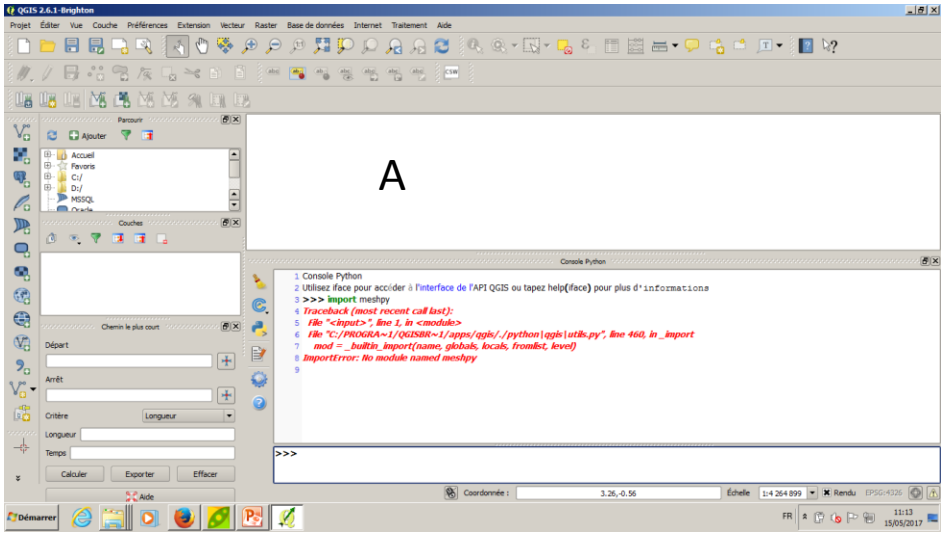
-Open the Python console

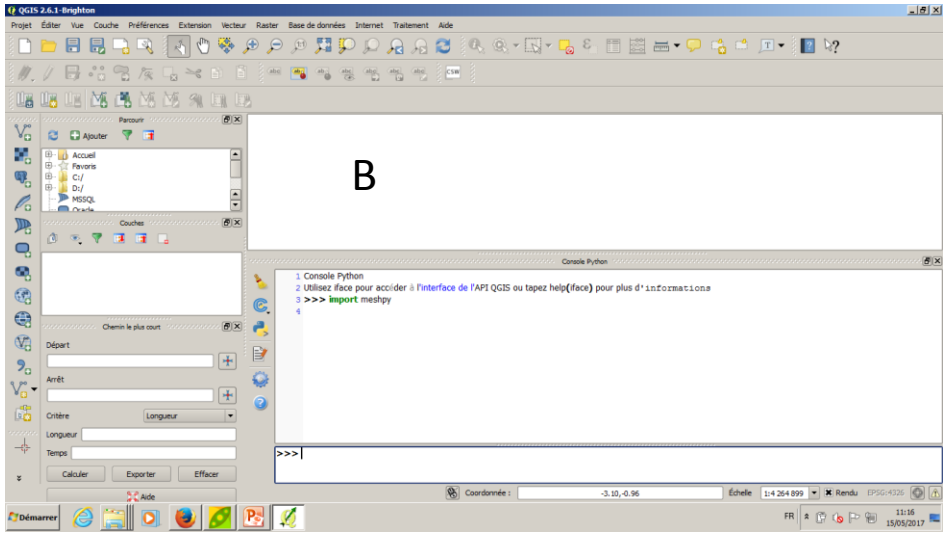
-Type: import meshpy

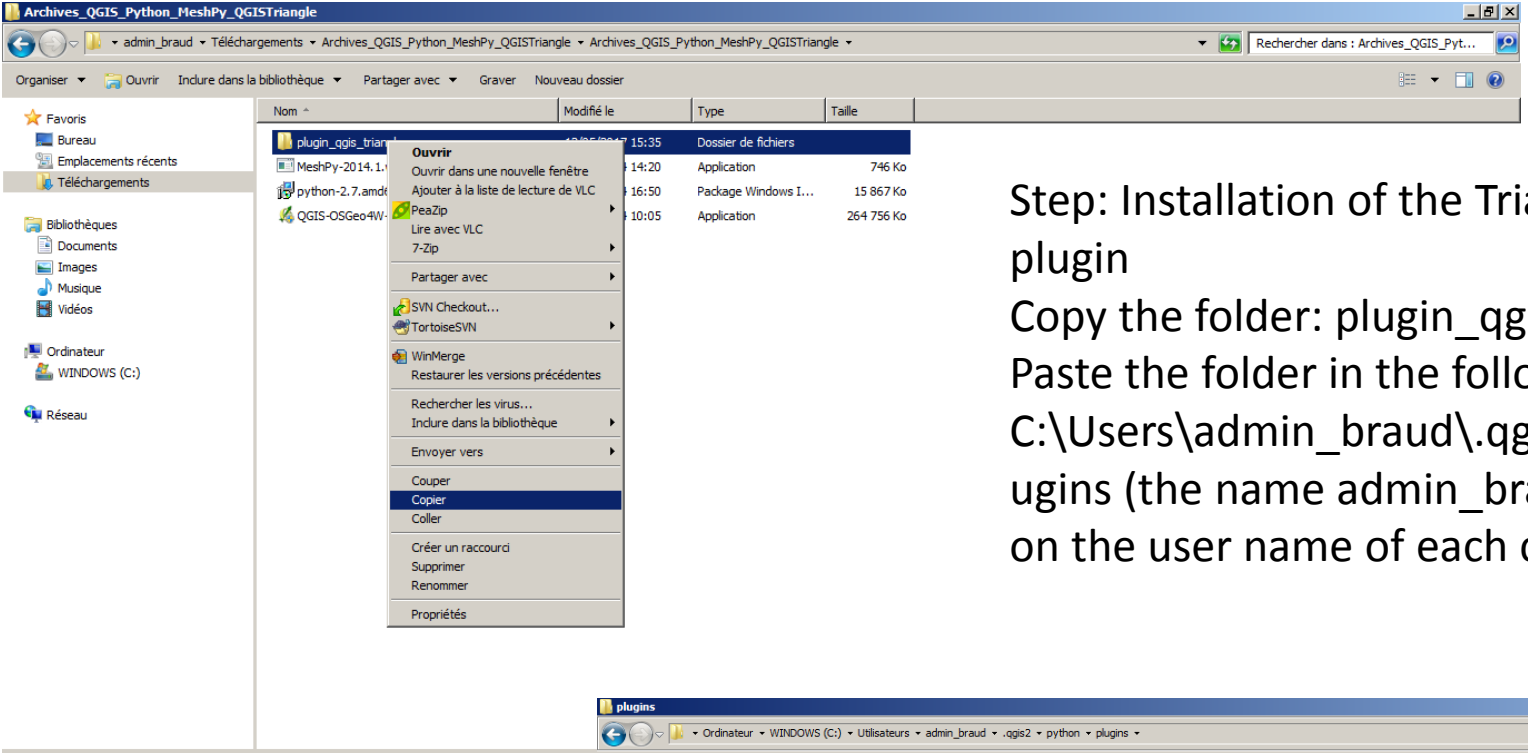
If the library is not well copied, the user gets Figure A;

If the library is well installed, the user gets Figure B

Once this step is finalized, QGIS must be closed to proceed with the installation of TriangleQGIS



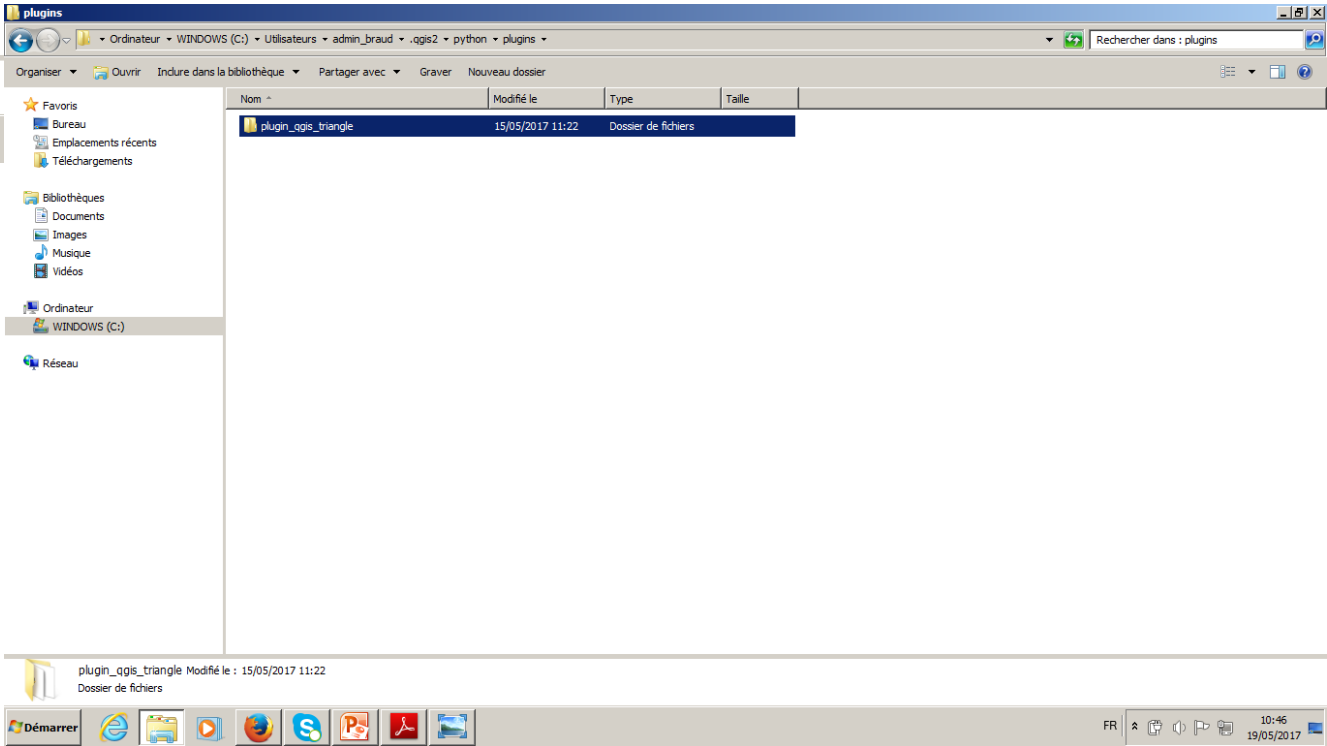


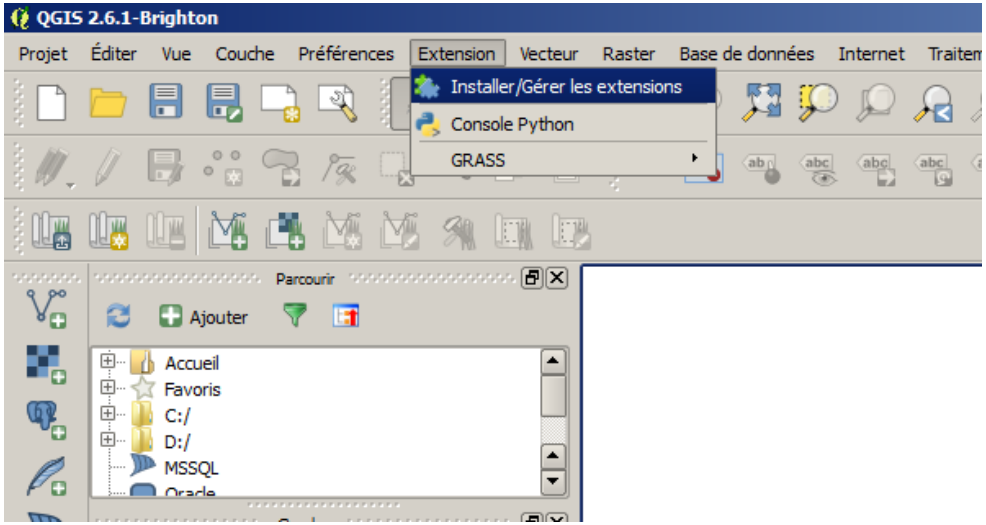


Step: Installation of the TriangleQGIS plugin

Copy the folder: plugin_qgis_triangle

Paste the folder in the following folder: C:\Users\admin_braud\.qgis2\python\plugins (the name admin_braud depends on the user name of each computer)



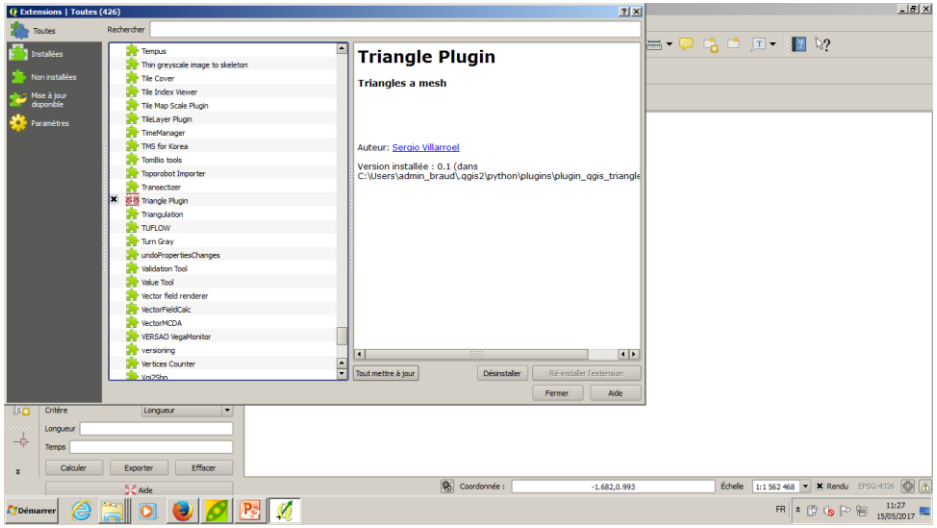


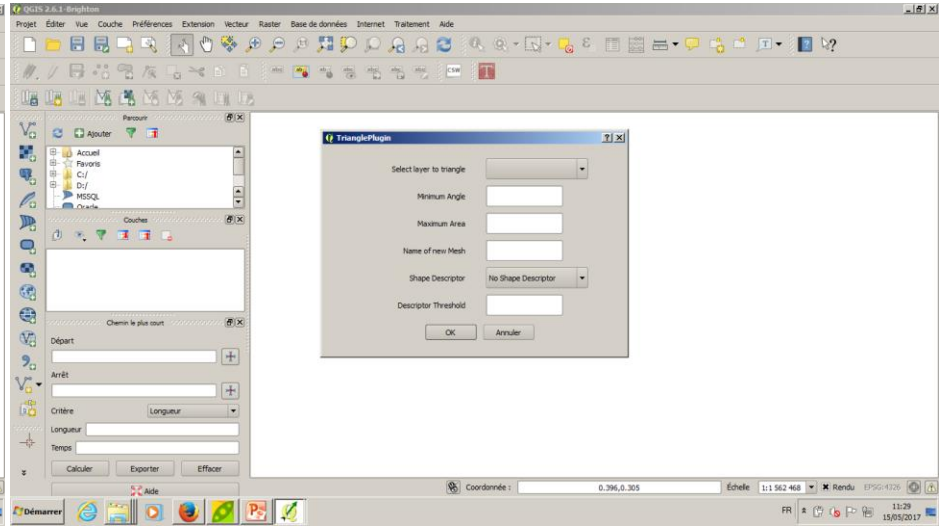
Step: Activation of the TriangleQGIS plugin

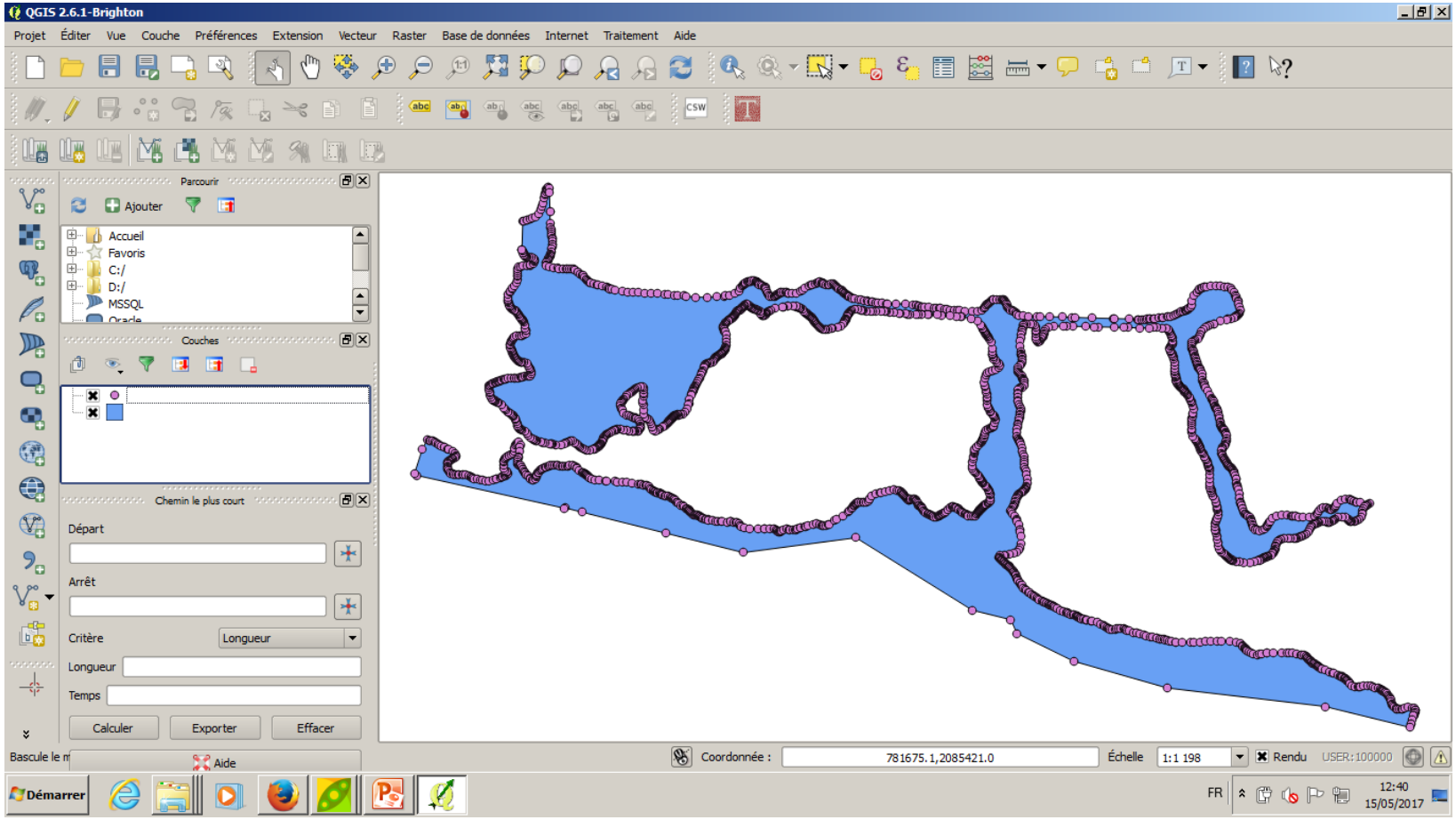
-Select Extension-> Install/Manage extensions

-Select Triangle with a click

Once installed, the Triangle icone appears as well as the interface to manage the the triangulation

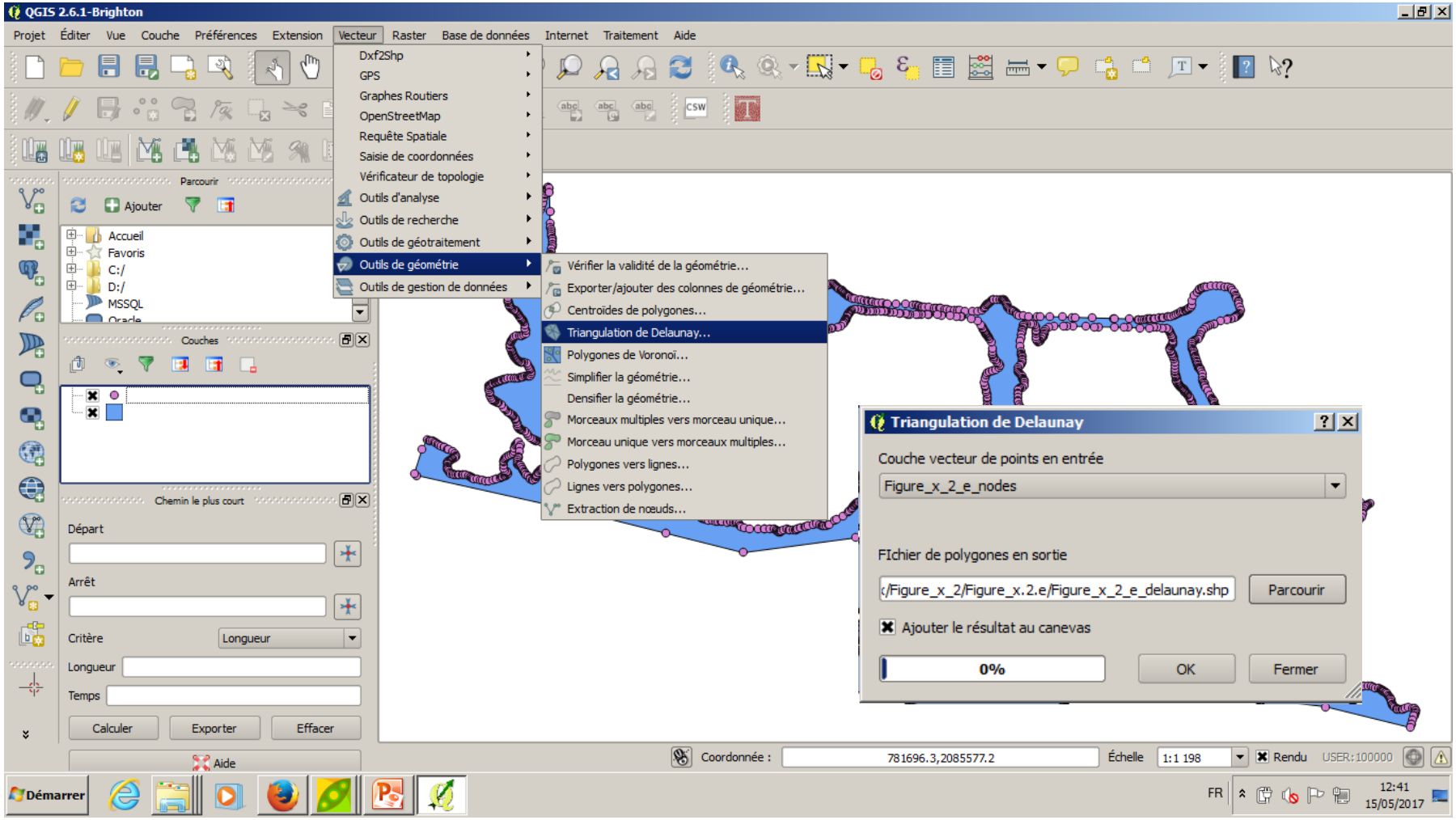






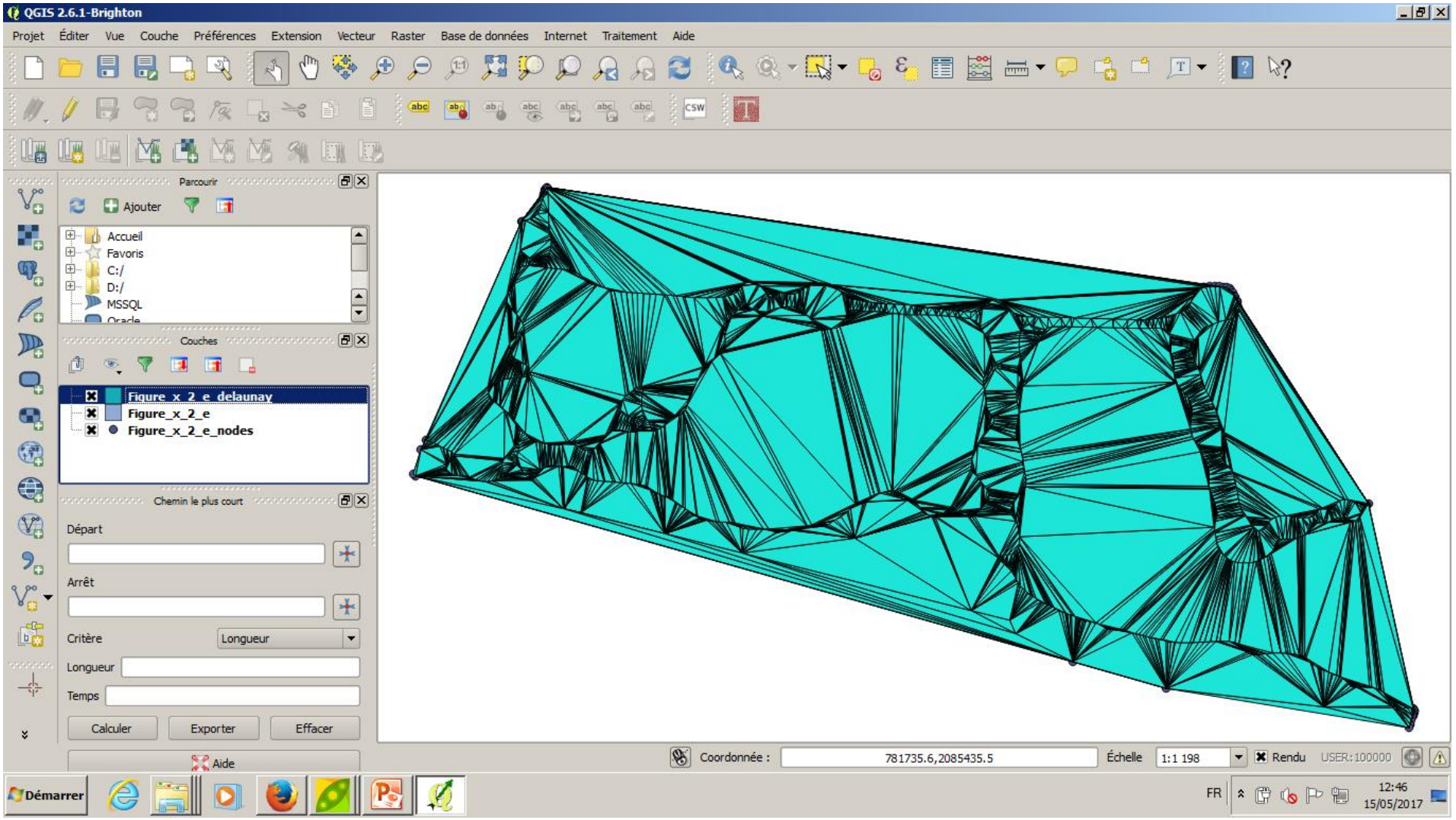
Step: Triangulation of a green area

-Load the shape Figure_x_2_e and Figure_x_2_e_nodes that is located in the folder: C:\Users\admin_braud\Downloads\Couches Vectorielles Chapitre x\Figure_x_2\Figure_x.2.e



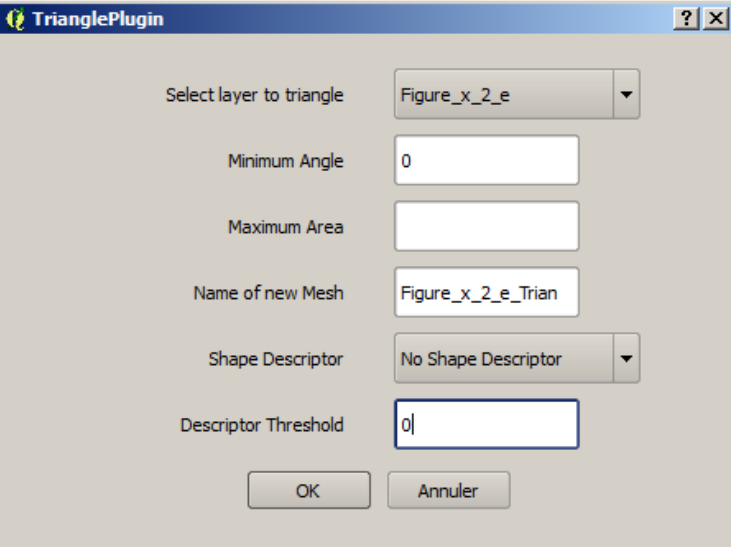
Step: Triangulation of a green area with the option Delaunay triangulation in QGIS

-Select the vectorial tool Vector-> Delaunay Triangulation; the input layer is Figure_x_2_e_nodes

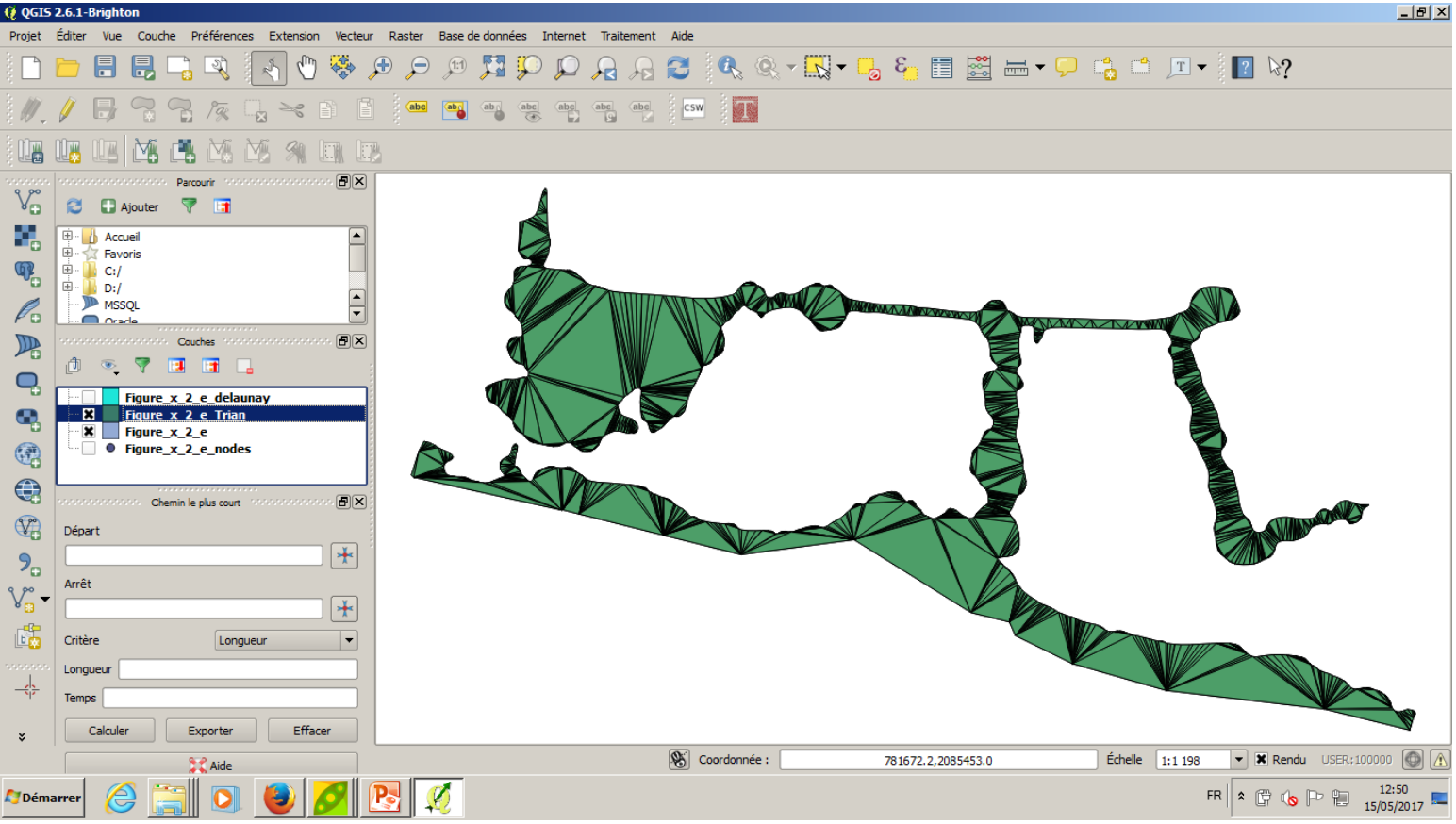


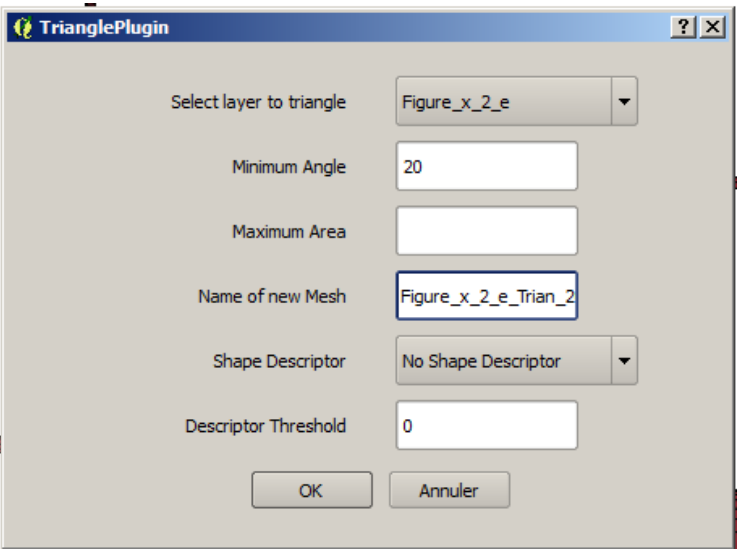
Step: Triangulation of a green area with the option Delaunay triangulation in QGIS (getting Figure x.5.b)

-The result of the Triangulation Delaunay option in figure (4667 triangles)



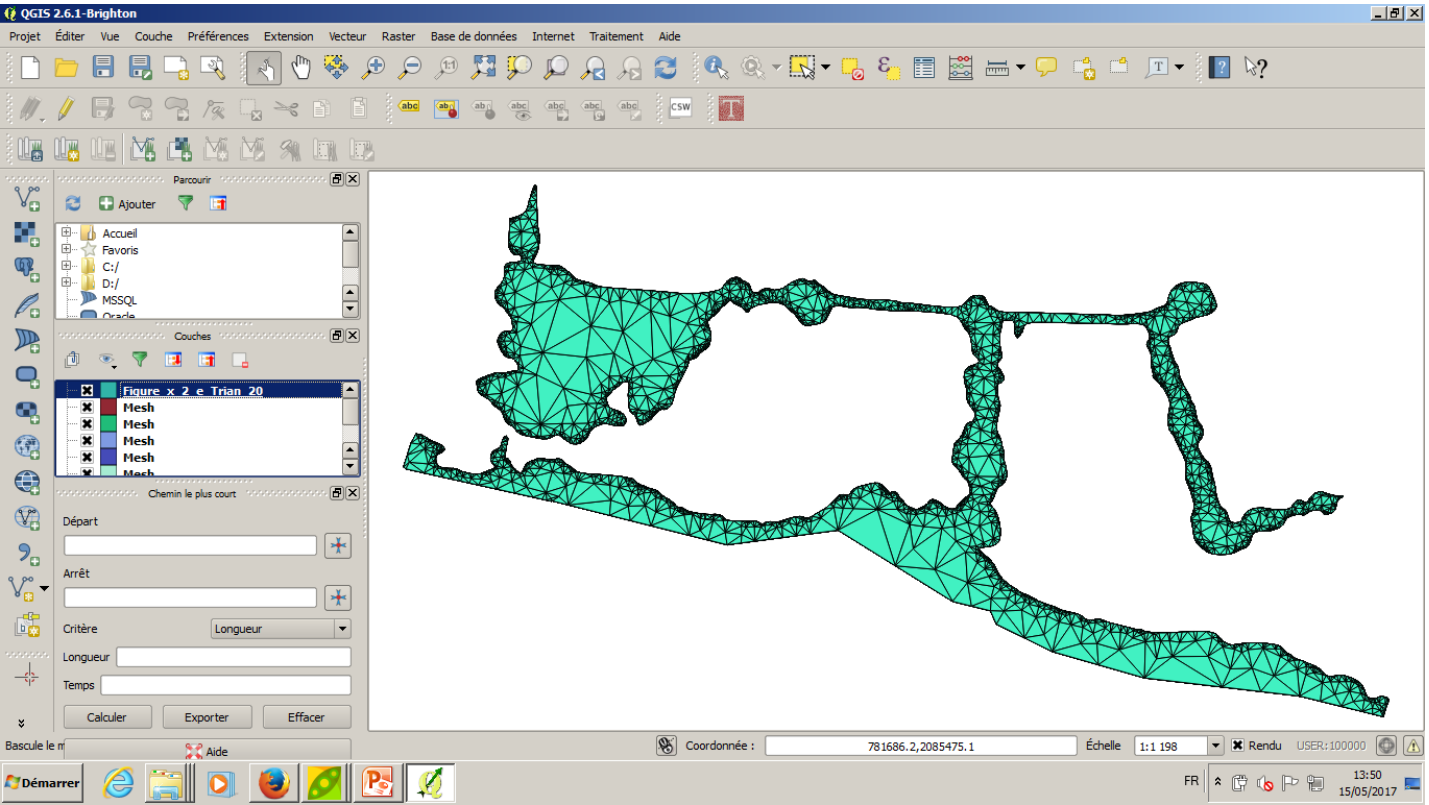
Step: Triangulation of a green area in Triangle (getting Figure x.5.c)
Option Delaunay Triangulation with constraints
-The triangulation of the green area generates triangles inside the polygon (2340 triangles) respecting the initial boundary of the initial polygon



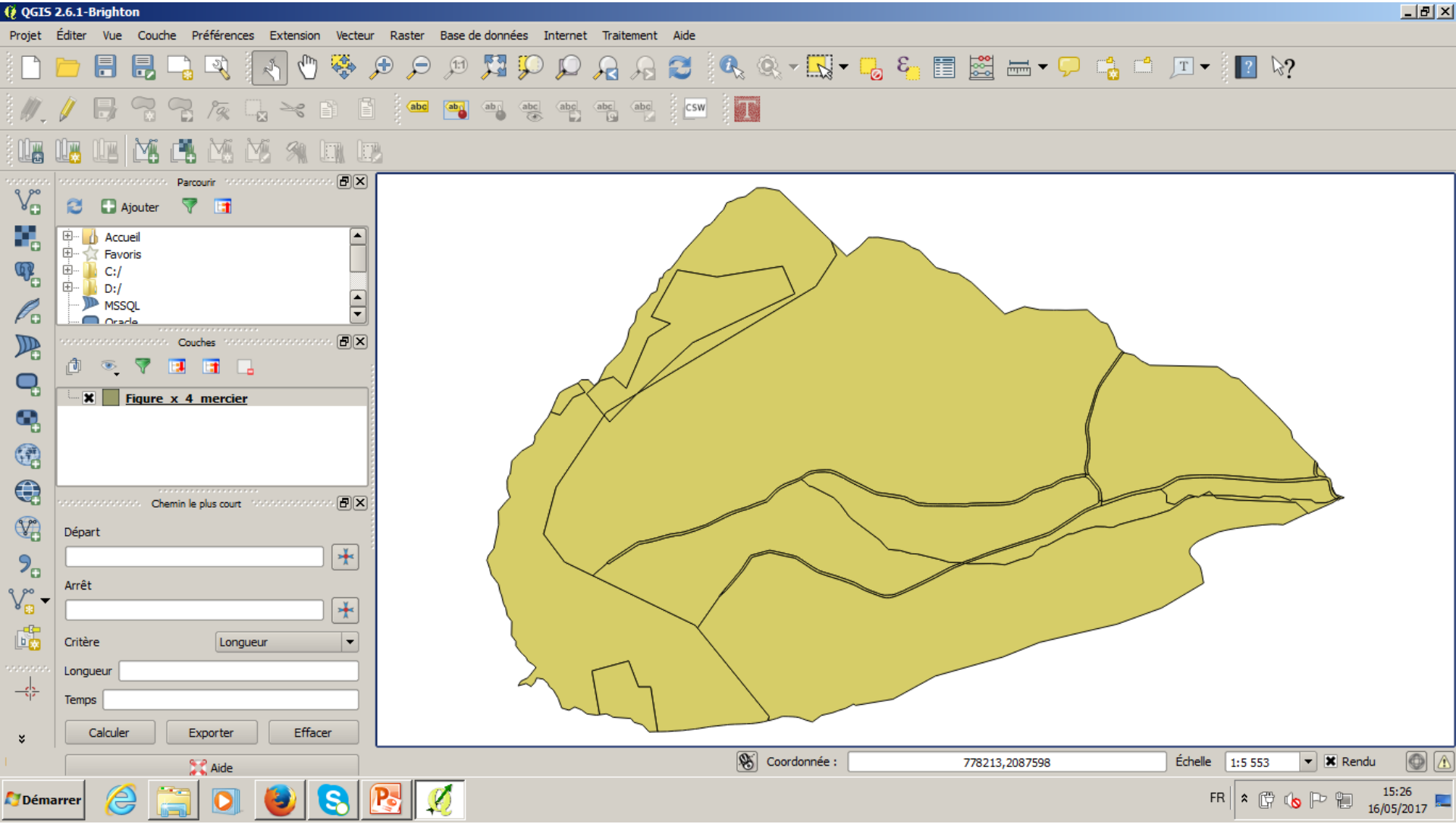


Step: Triangulation of a green area in Triangle (getting Figure x.5.d) with option conformal Triangulation (Maximum angle30°)

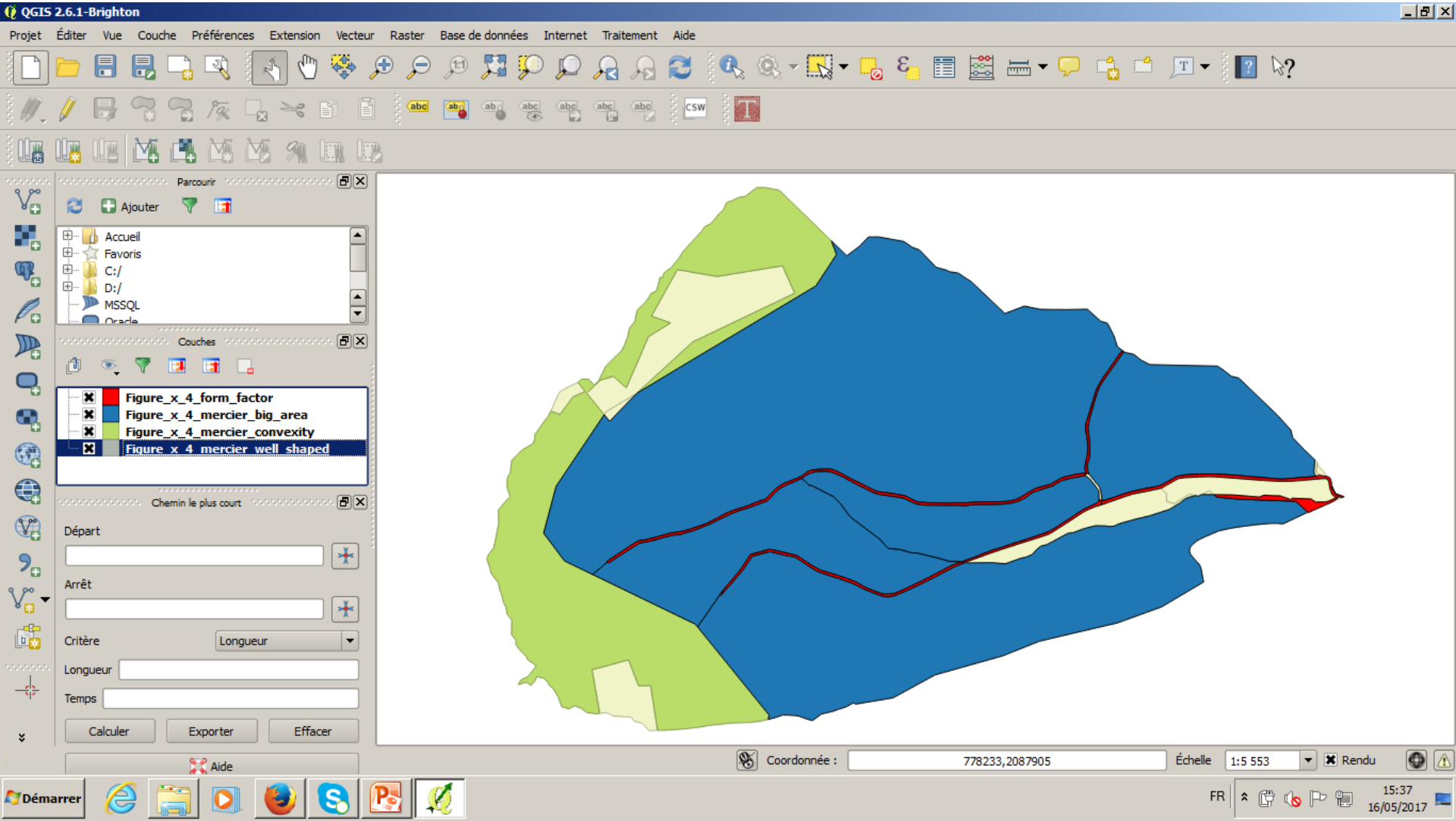
- The triangulation of the green area generates triangles inside the polygon (5247 triangles) with a restriction on the interior angle of 20° maximum



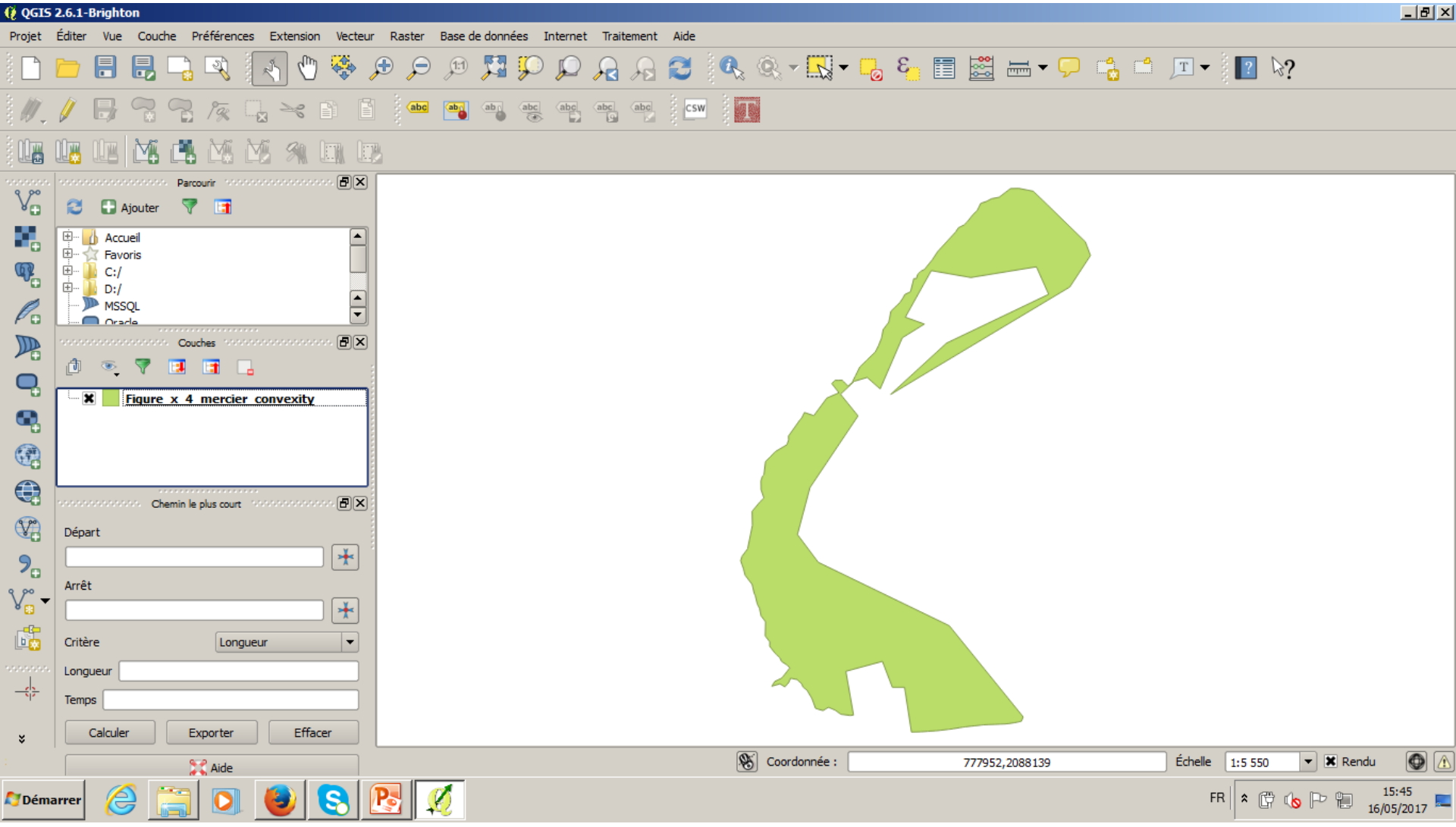
Step: Triangulation of a periurban catchment: the Mercier catchment
The corresponding .shape, Figure_x_4_mercier.shp, is located int he folder
C:\Users\admin_braud\Downloads\Couches Vectorielles Chapitre x\Figure_x_4
It shows a periurban sub-catchment composed of 23 elements



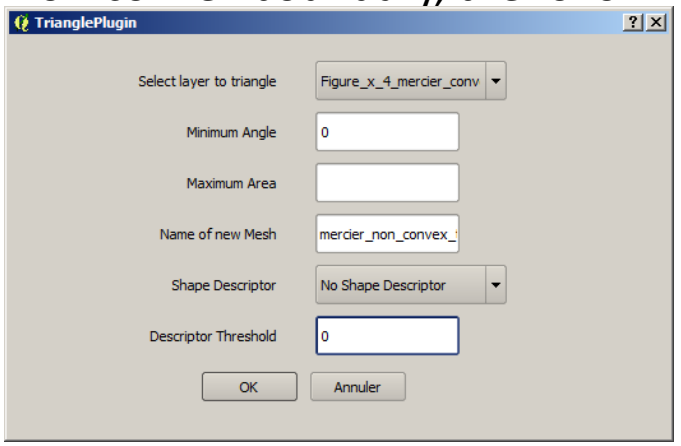
Step: Selection of bad-shaped irregular polygons without shape format (cf Figure x.4)
Polygons of the Mercier are divided into four groups:
Layer Figure_x_4_mercier_convexity.shp: shows the polygons with a convexity <0,75
Layer Figure_x_4_form_factor.shp : shows the polygons with a form factor <0,20
Layer Figure_x_4_mercier_big_area.shp : shows the elements with area > 2 ha
Layer Figure_x_4_mercier_well_shaped.shp : shows the elements without shape problem



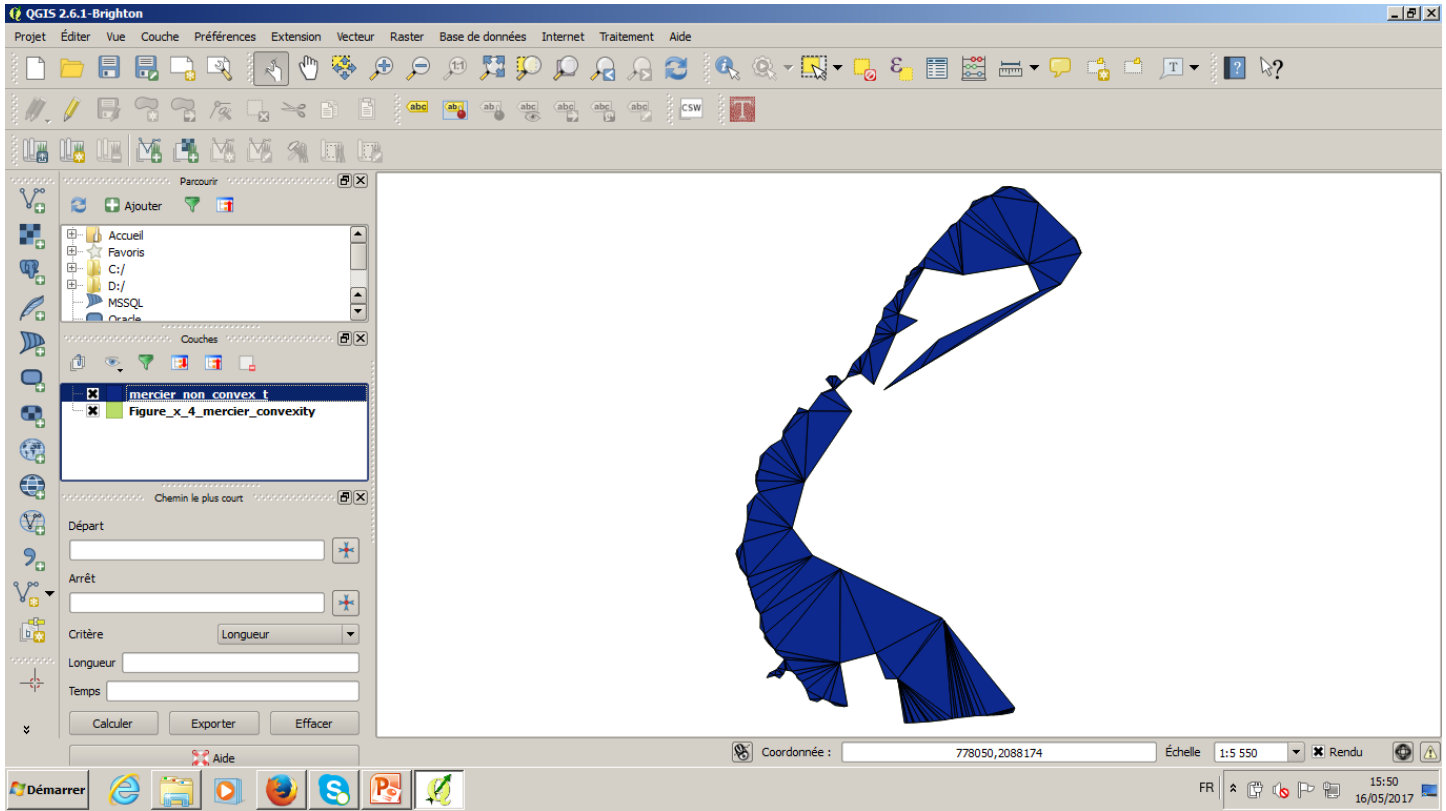
Step: Triangulation of non-convex elements (Getting figure x.30.a)
-Load the .shp: Figure_x_4_mercier_convexity.shp



Step: Triangulation of non-convex elements (Getting figure x.30.b)
-load the .shp: Figure_x_4_mercier_convexity.shp
-to get a triangulation that preserve the initial non-convex boundary, the following options are used:



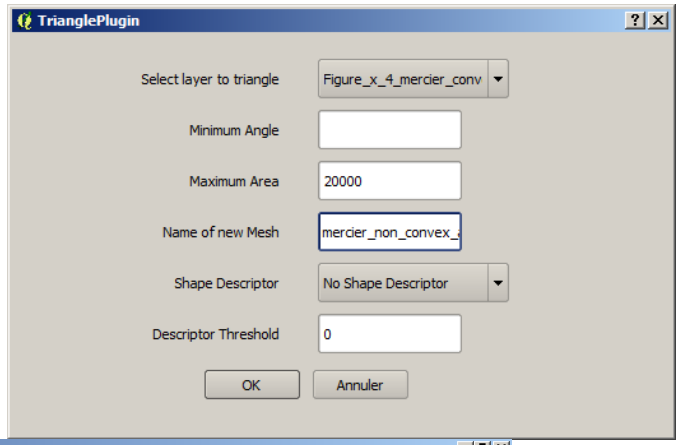
This figure is part
Figure x.33
Non-convex element

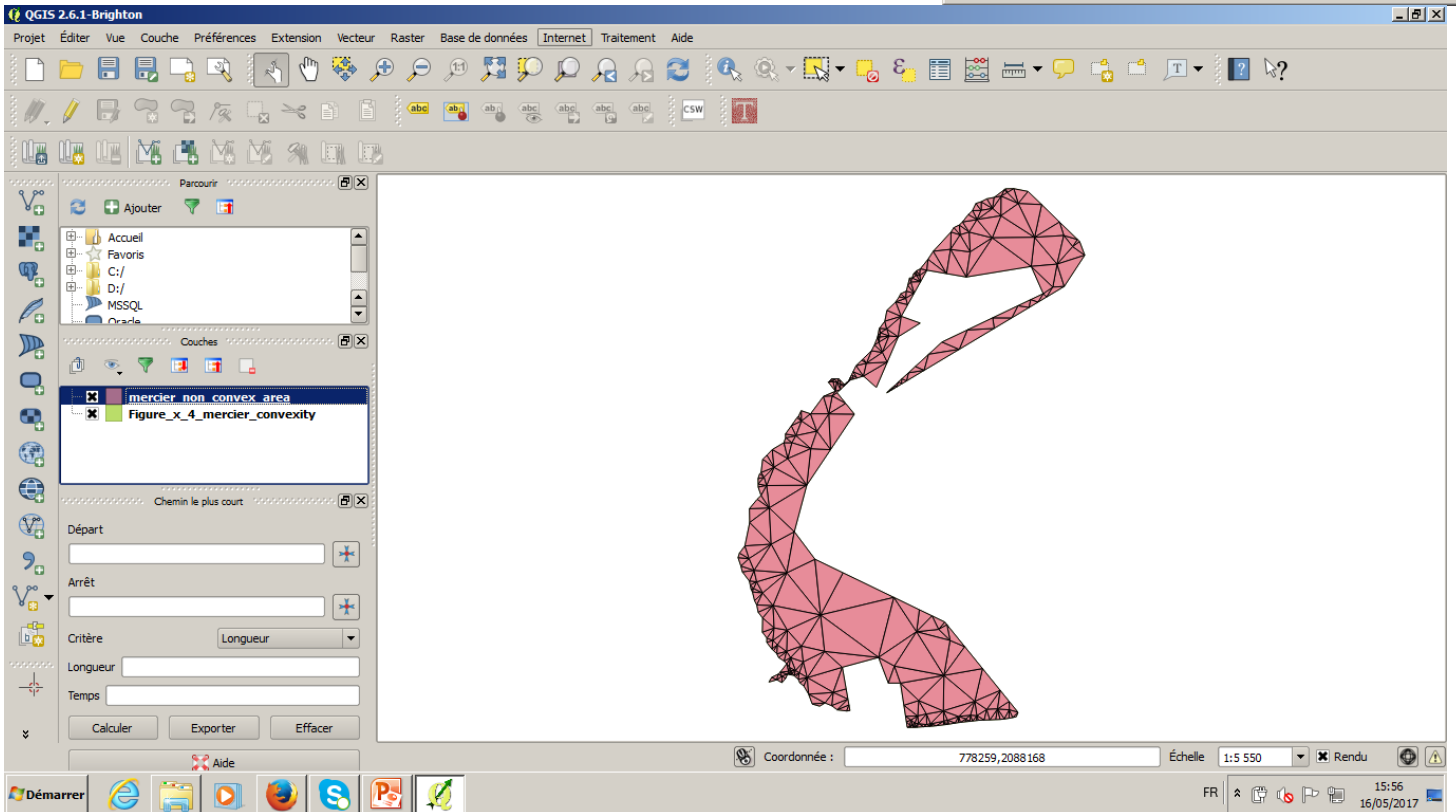


Step: Triangulation of non-convex elements (Getting figure x.30.c)

-Load the .shp: Figure_x_4_mercier_convexity.shp

- To get a triangulation that only generates triangles with an area larger than 20000 m2 the following options are used:

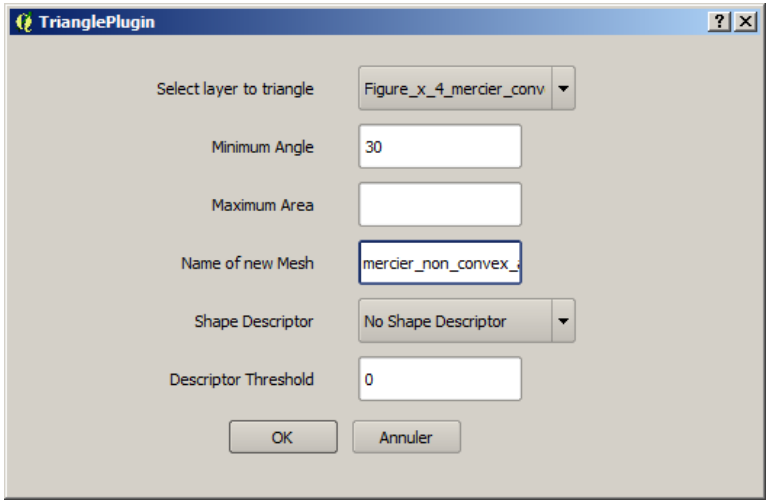


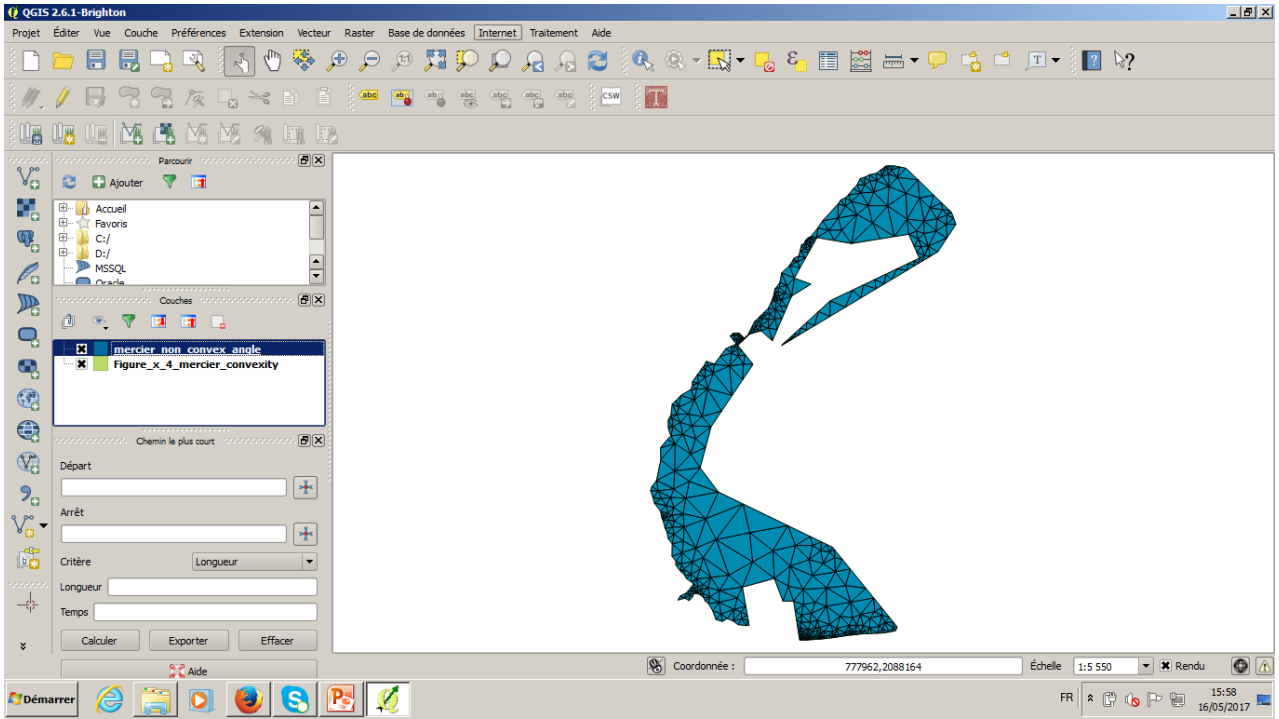


Step: Triangulation of non-convex elements (Getting figure x.30.d)

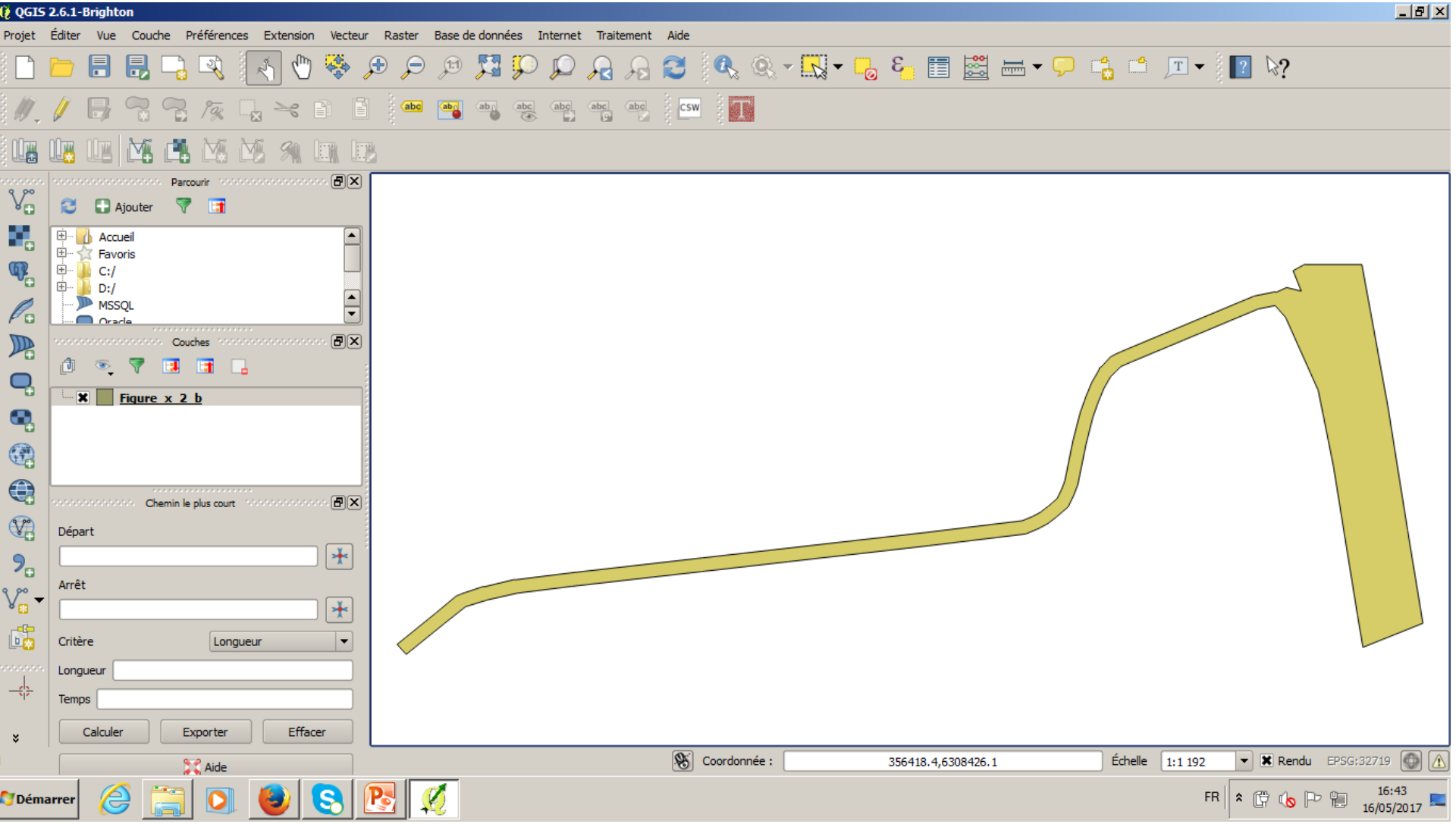
-Load the .shp: Figure_x_4_mercier_convexity.shp

-to get the triangulation that only contains triangles with angles lower than 30°, the following options are used:





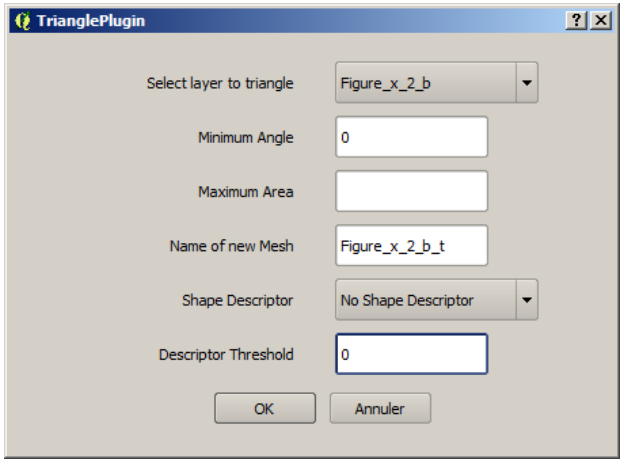
Step: Triangulation of a long and thin element (Getting Figure x.31.a)
-Load the .shp: Figure_x_2_b.shp

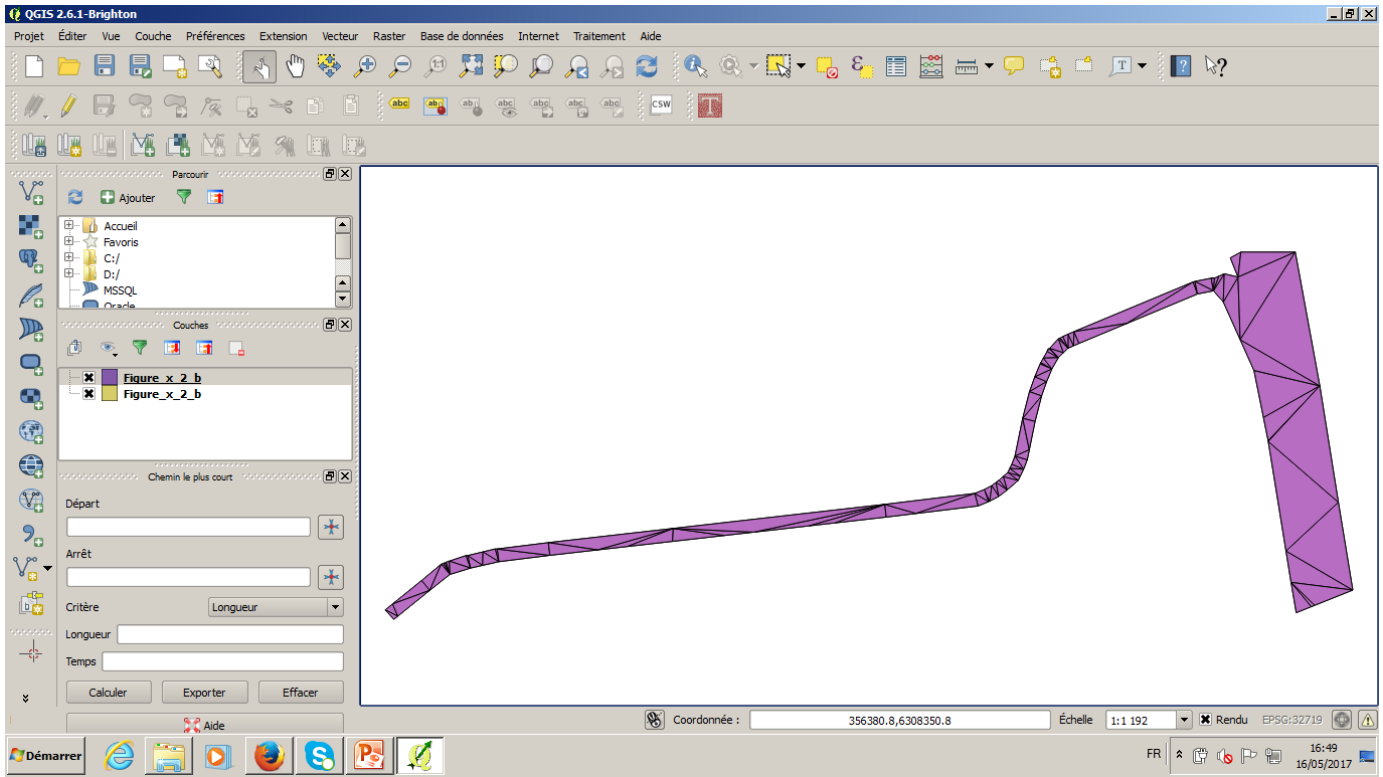


Step: Triangulation of a long and thin element (Getting Figure x.31.b)

-Load the .shp: Figure_x_2_b.shp

- to get a triangulation that preserve the initial polygon boundary, the following options are used :

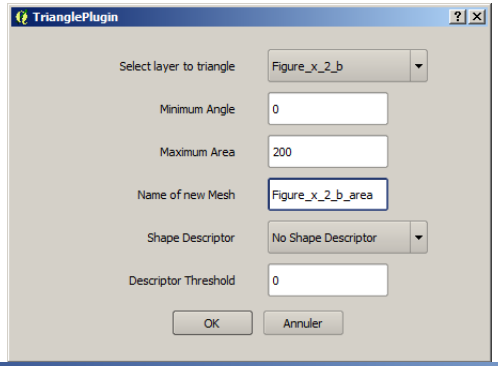


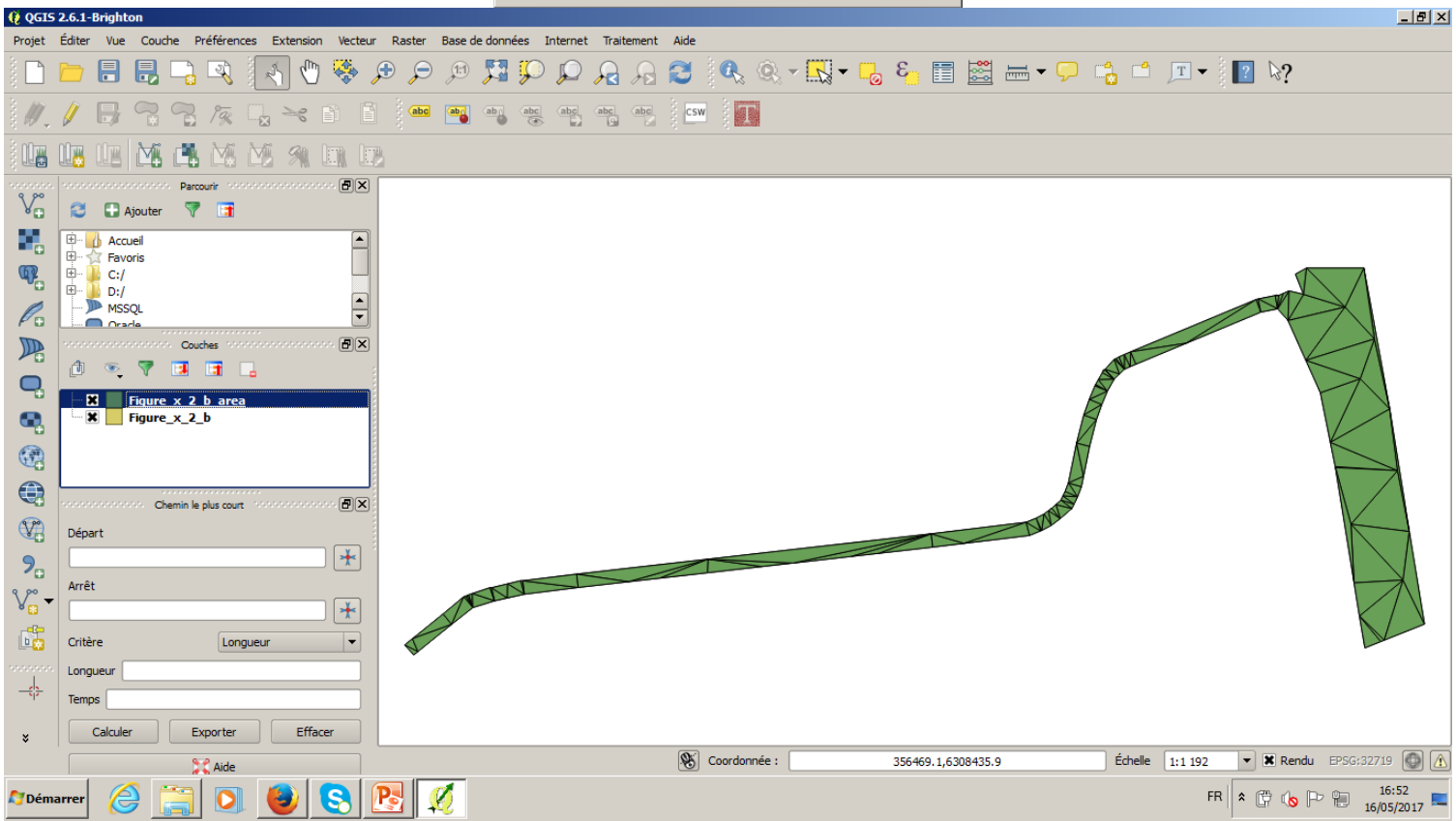


Step: Triangulation of a long and thin element (Getting Figure x.31.c)

-Load the .shp: Figure_x_2_b.shp

-To get a triangulation that only generates triangles with an area larger than 200 m2 the following options are used:

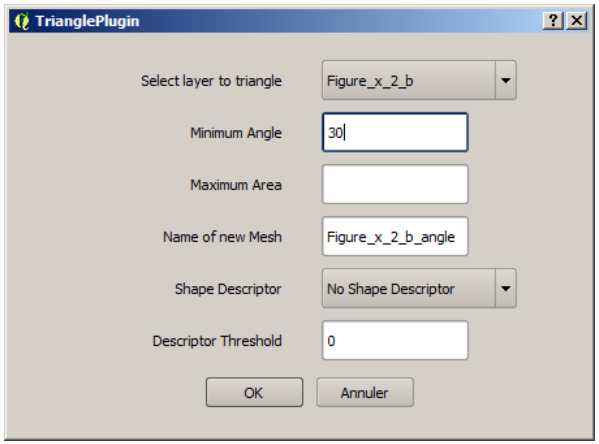


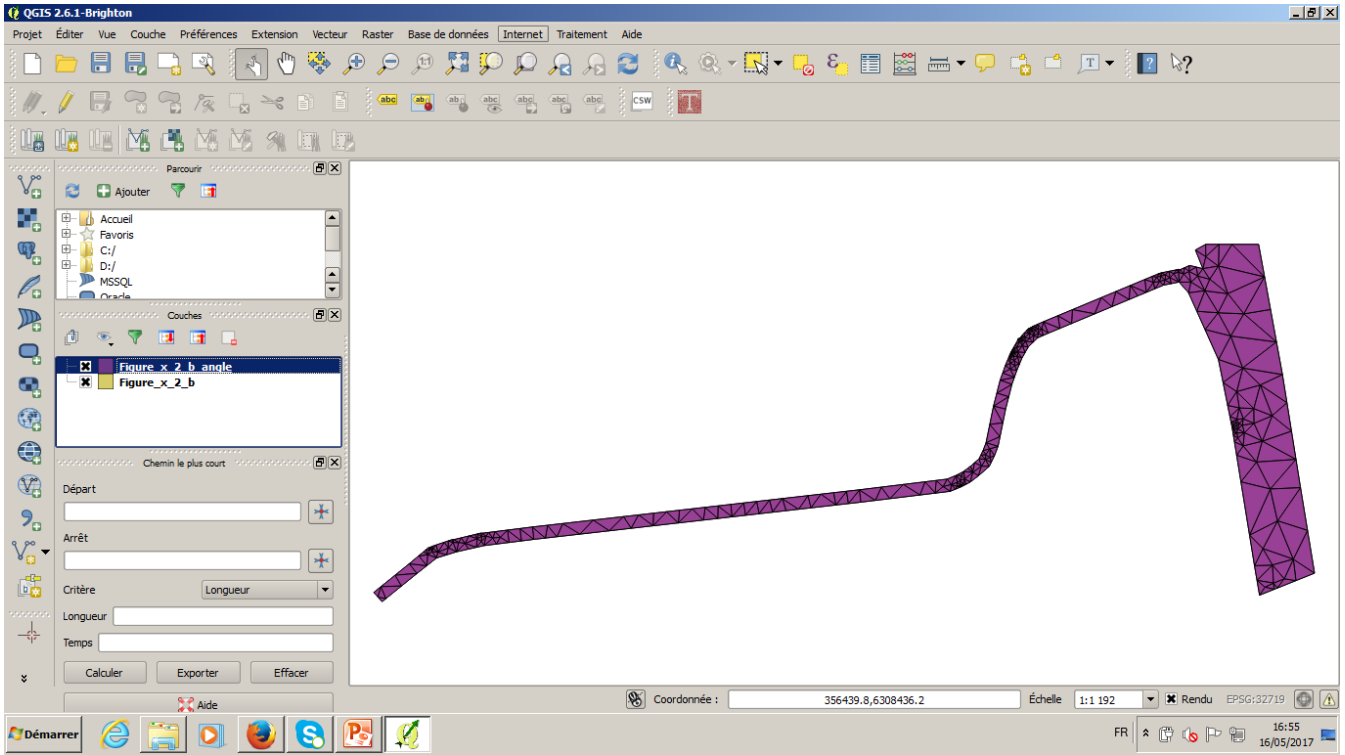


Step: Triangulation of a long and thin element (Getting Figure x.31.d)

-Load the .shp: Figure_x_2_b.shp

- to get the triangulation with angles restriction of 30°, the following options are used:

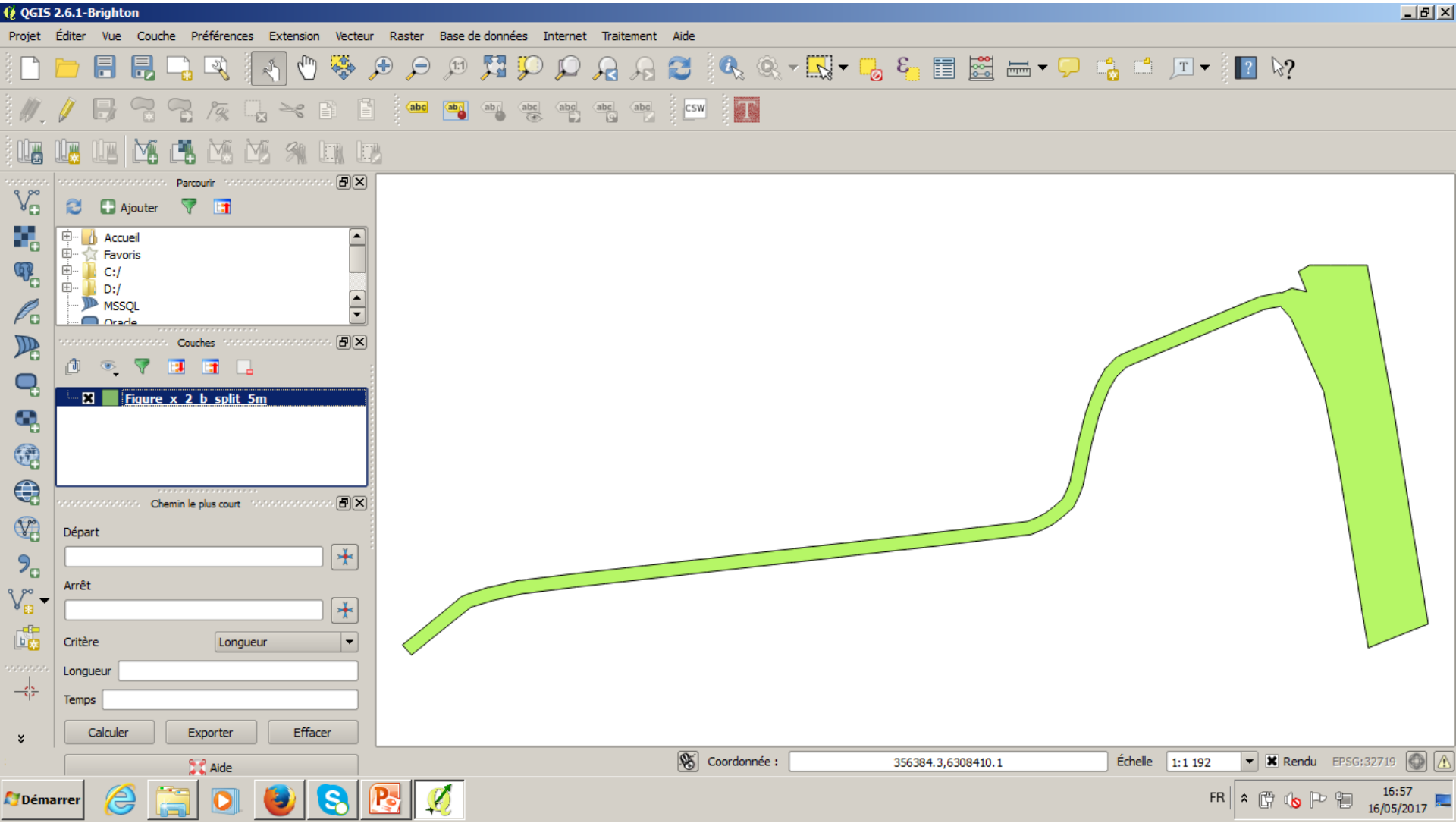




Step: Triangulation of a long and thin element with nodes separated by a maximum distance of 5 m (Getting Figure x.31.e)

-Load the .shp: Figure_x_2_b_split_5m.shp (The .shp where additional points are already inserted is given. To do that, the user will have to use the v.split function from Grass (see p.32 of the Geo-PUMMA Users' Manual)

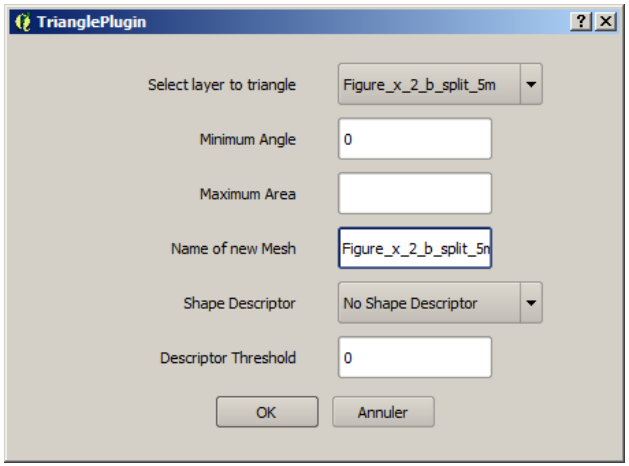
-to get the triangulation with angles restriction of 30°, the following options are used:

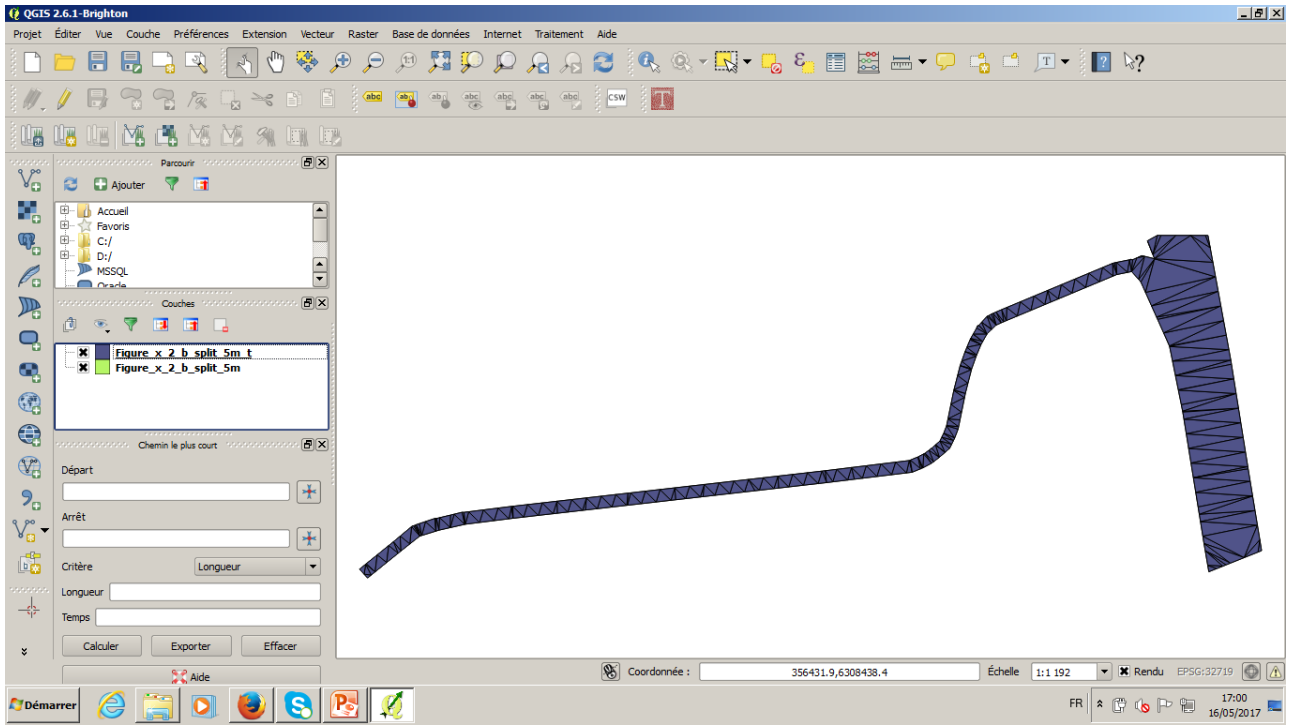


Step: Triangulation of a long and thin element (Getting Figure x.31.f)

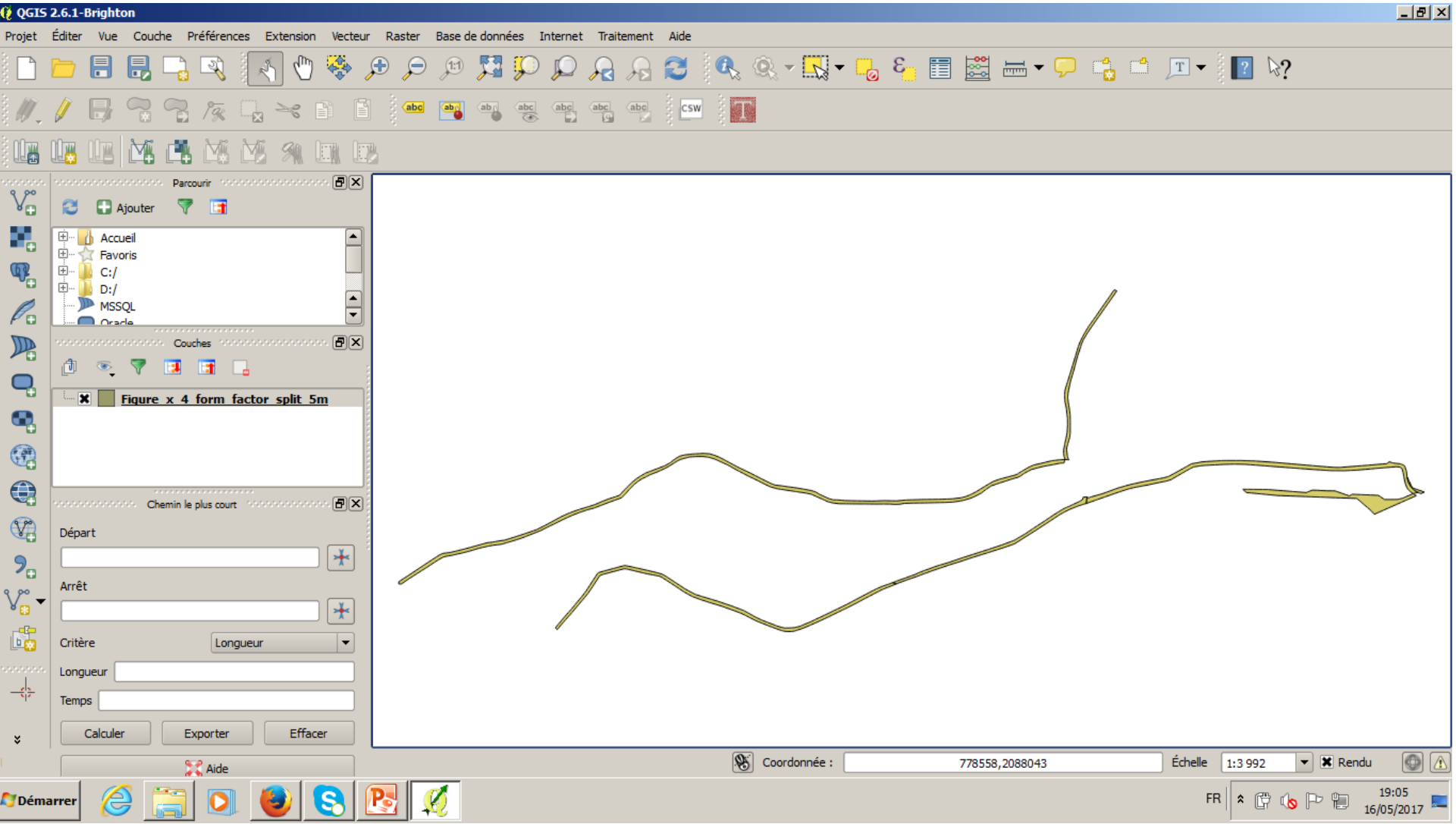
-Load the .shp: Figure_x_2_b_split_5m.shp

-to get a triangulation without restriction on surface and angle, use the following options:

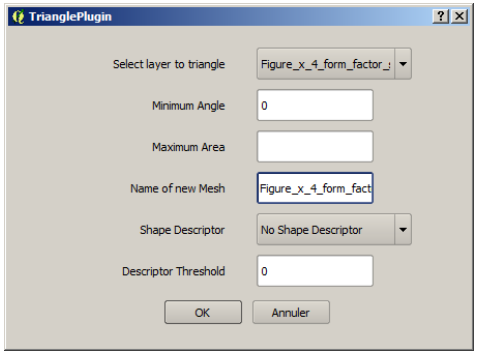


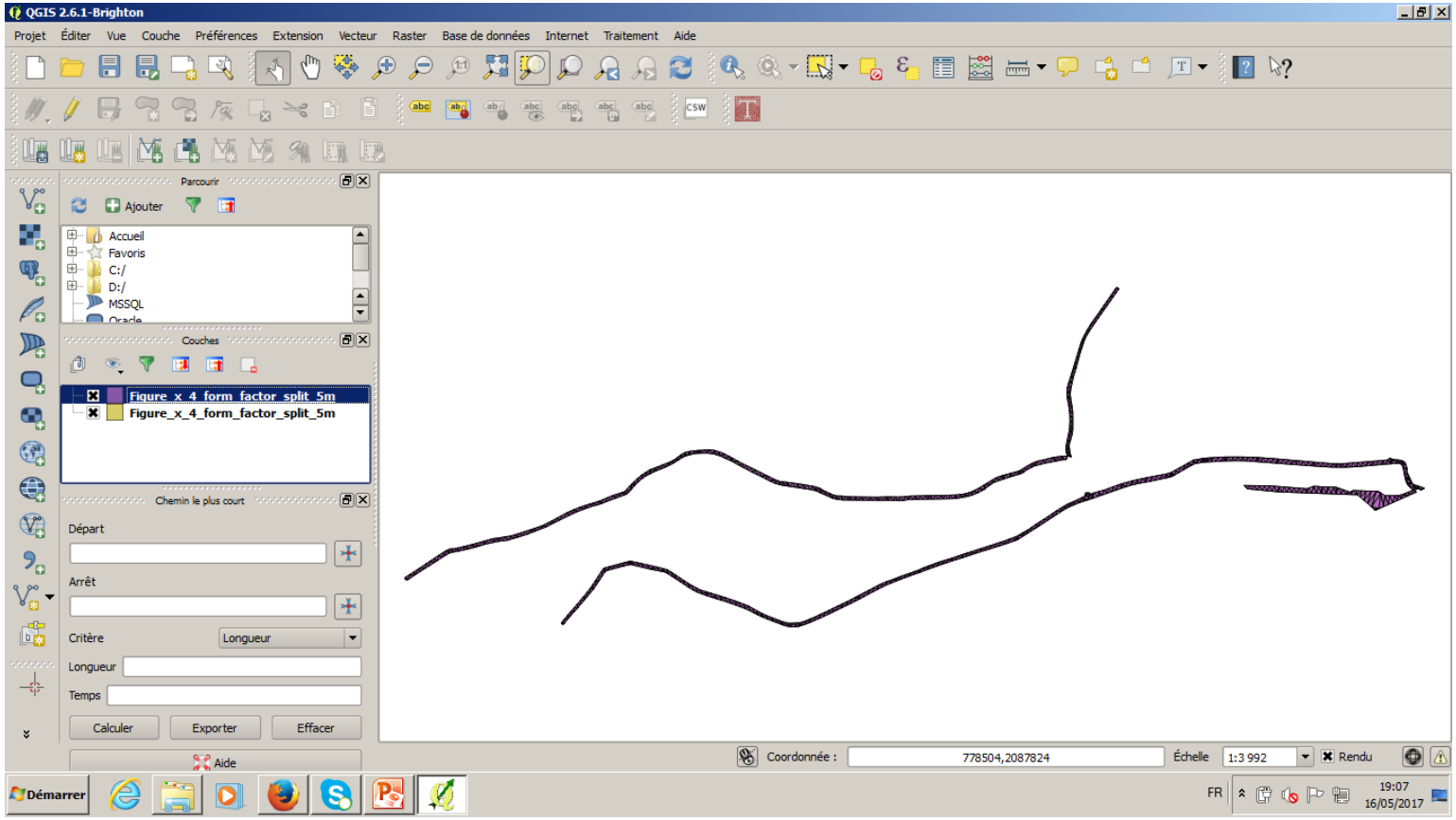


Step: Triangulation of a long and thin element from the Mercier
-Load the layer Figure_x_4_form_factor_split_5m.shp that is located in the folder:
C:\Users\admin_braud\Downloads\Couches Vectorielles Chapitre x\Figure_x_4



Step: Triangulation of a long and thin element from the Mercier
-to get a triangulation without restriction on area and angle, use the following options:
This figure is part of
Figure x.33
Long and thin element





Step: Triangulation of a polygon with a too large area

-Load the .shp: Figure_x_4_mercier_big_area.shp

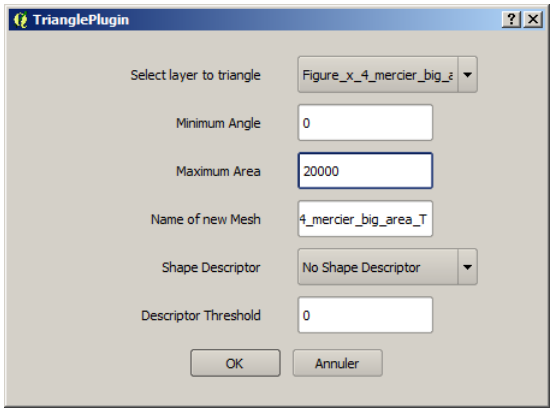
-to get a triangulation with area restriction of 2ha, use the following options:

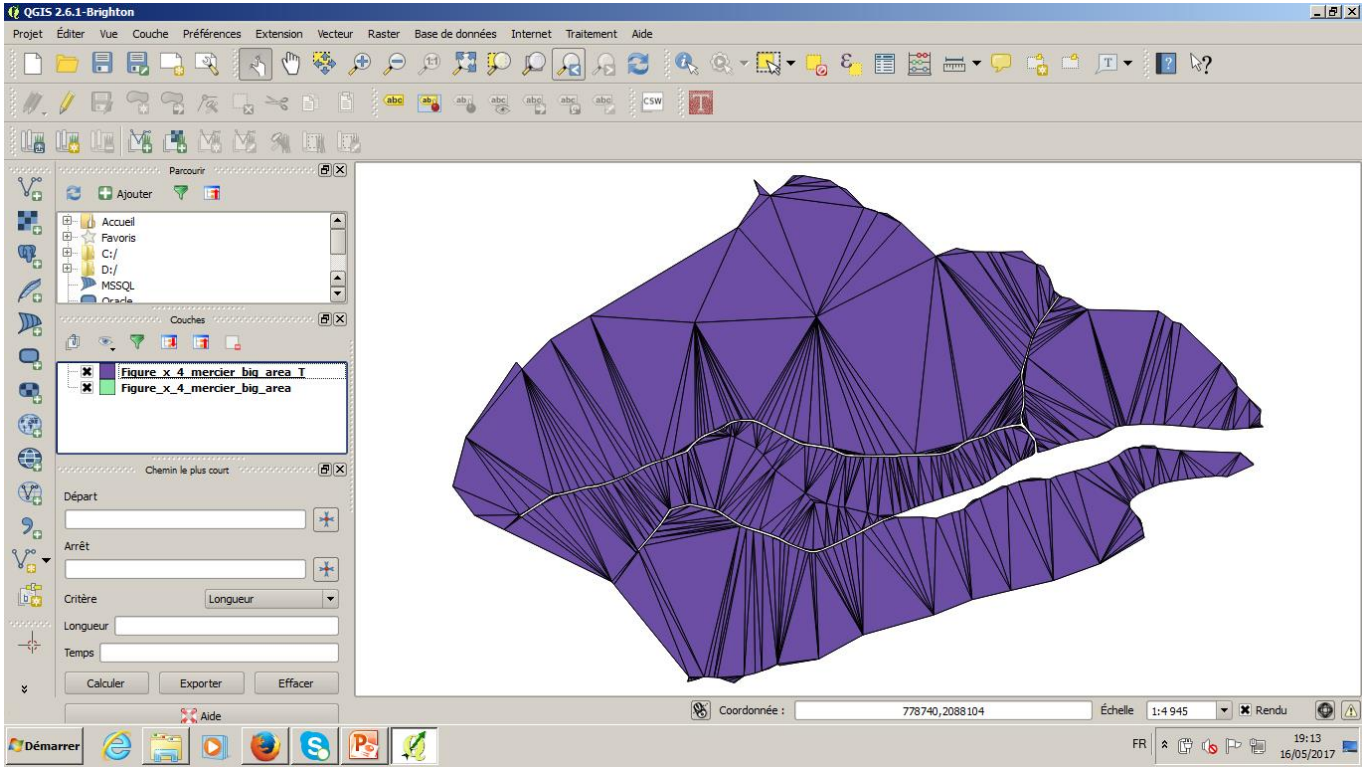
This figure is part of

Figure x.33

Too large element

Restriction on the area





Step: Triangulation of the Mercier mesh

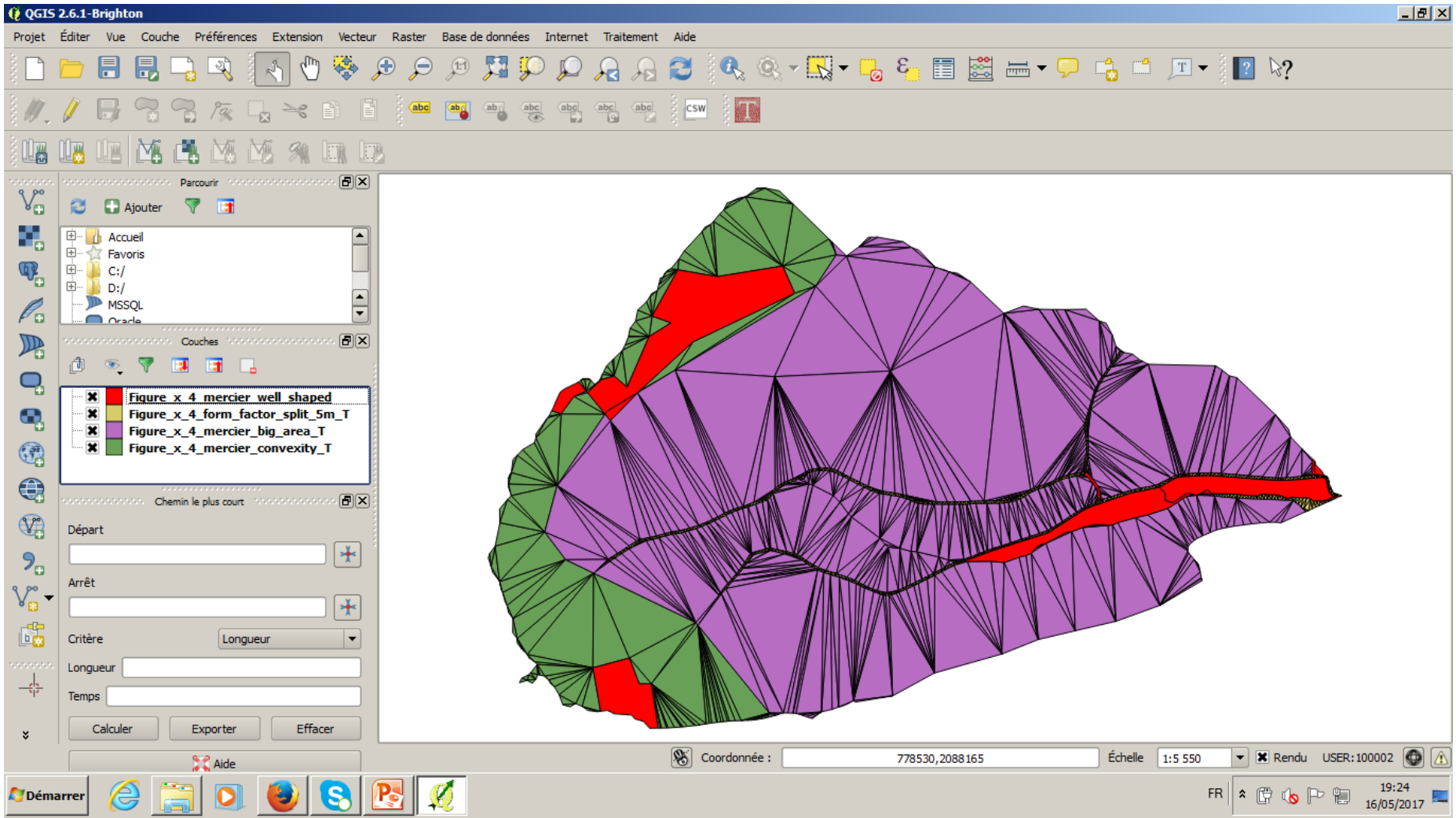
The following GIS layers are available: (Figure x.33)

-Figure_x_4_mercier_big_area_T

-Figure_x_4_form_factor_split_5m_T

-Figure_x_4_mercier_convexity_T

-Figure_x_4_mercier_well_shaped



Part B: (QGIS-GRASS-Virtual box)

Dissolution of the triangulation and application to the Mercier catchment

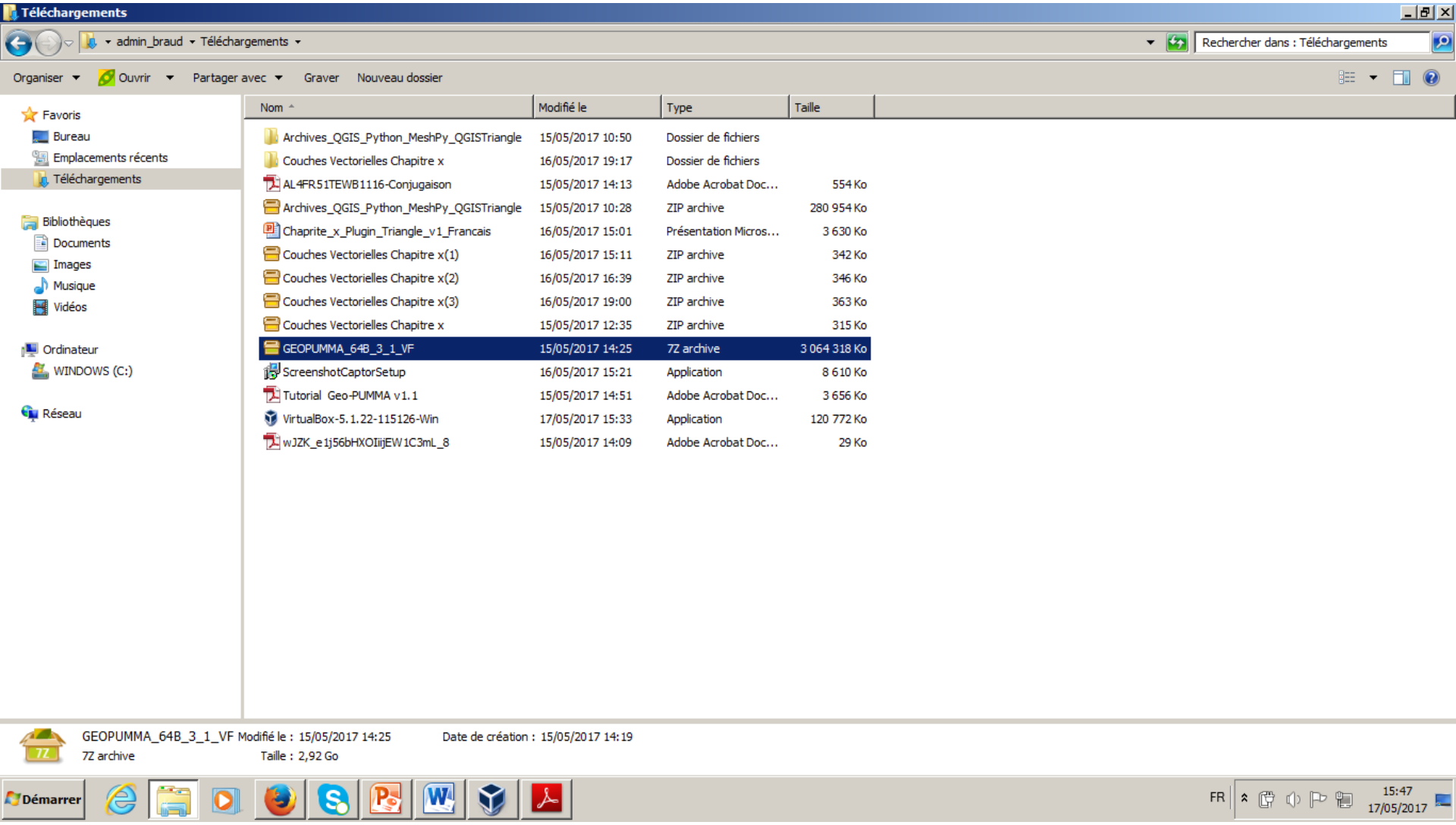
Step: download the GeoPUMMA Virtual box
-This downloading can be performed on the following web site:
http://DOI.org/10.5281/zenodo.821563
And for the source codes
https://forge.irstea.fr/projects/geopumma

If more information is required, the user can download the detailed tutorial Geo-PUMMA v1.1.pdf:
https://forge.irstea.fr/projects/geopumma/files

Step: Download the GeoPUMMA virtual box

-If the download was successful, the following file should be available in the download folder

-Then the virtual box, available in the .7zip archive must be decompressed

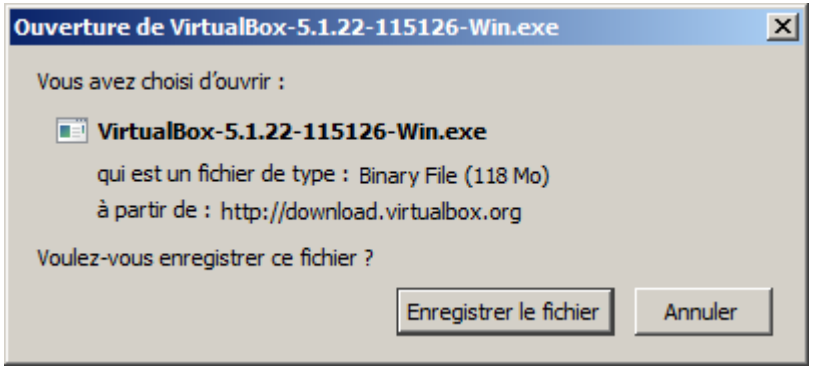


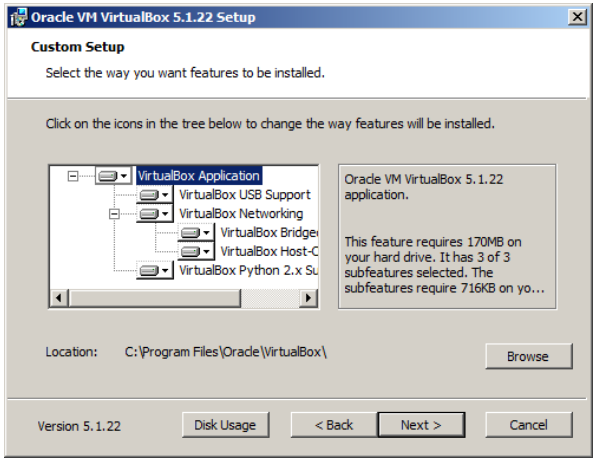
Step: Installation of the Virtual box
Select Downloads

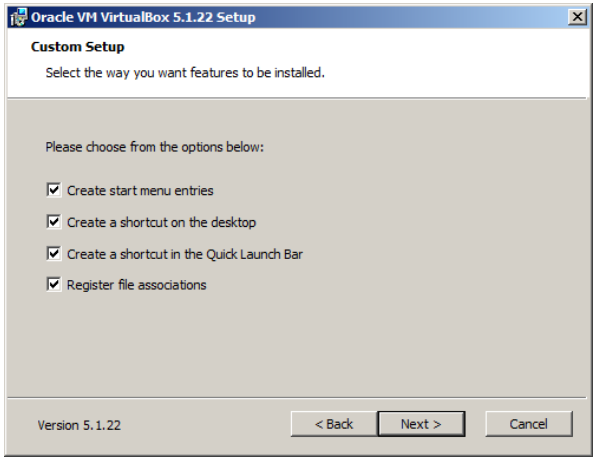
Step: Installation of the Virtual box
-Download the version for Windows

Step: Installation of the Virtual box

-Select « Save » and accept all the default installation options in all the windows that wil open later:

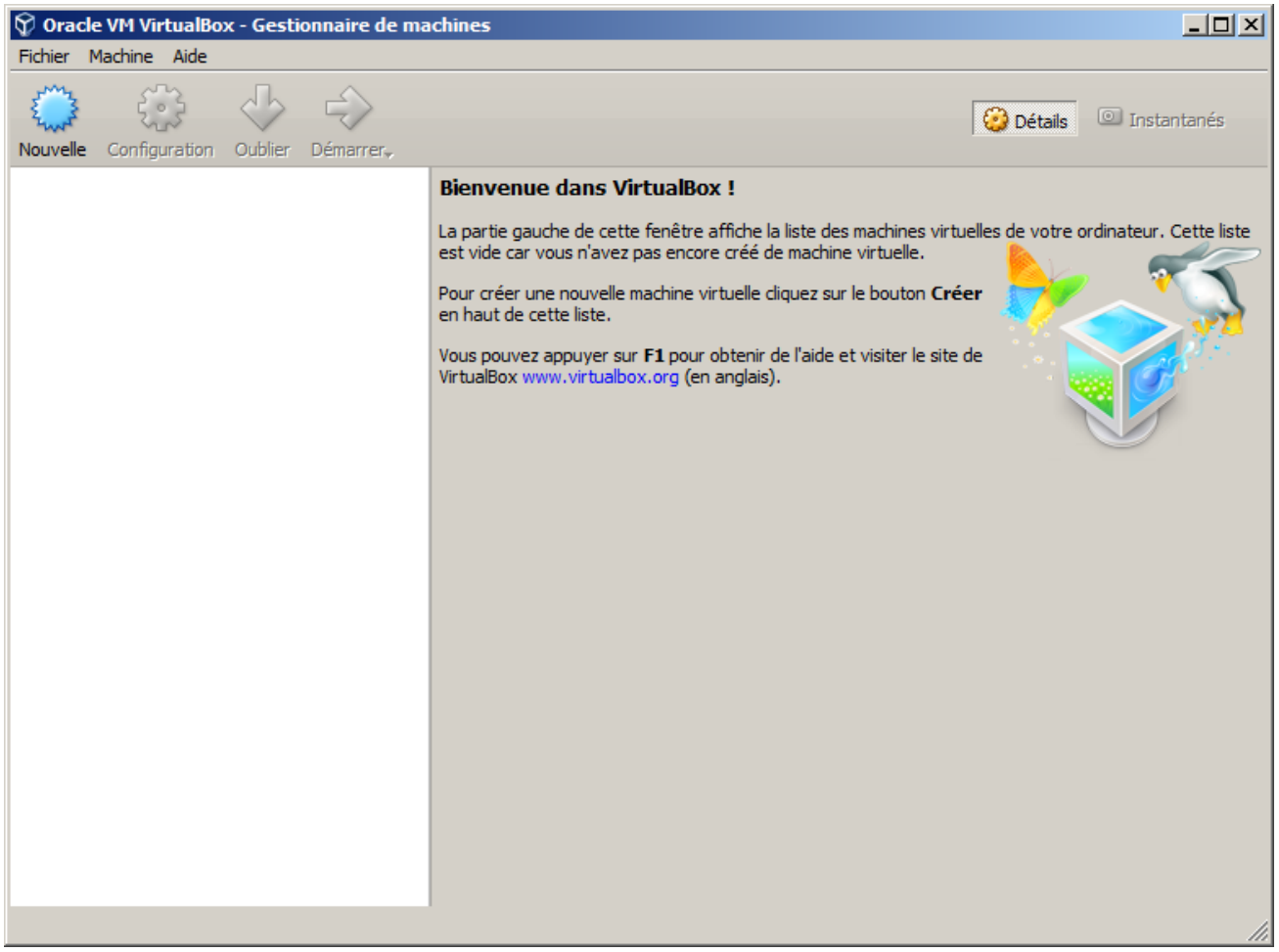




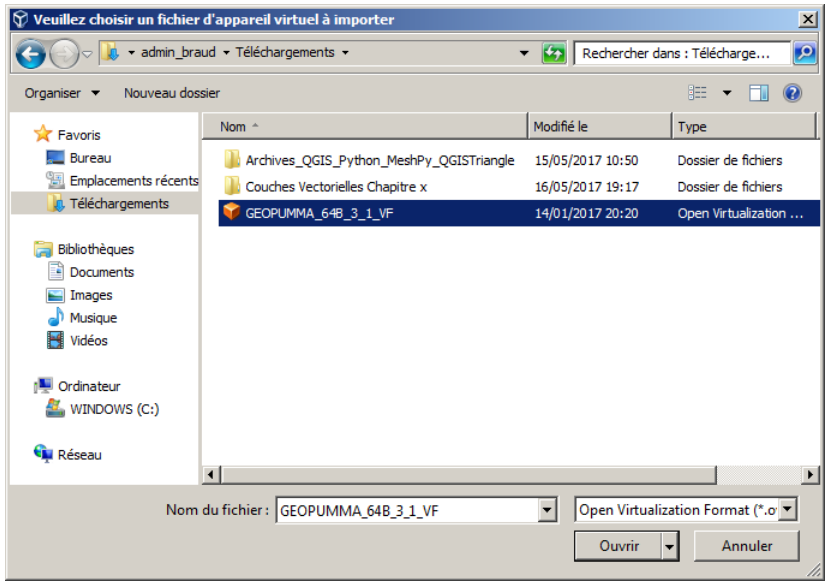


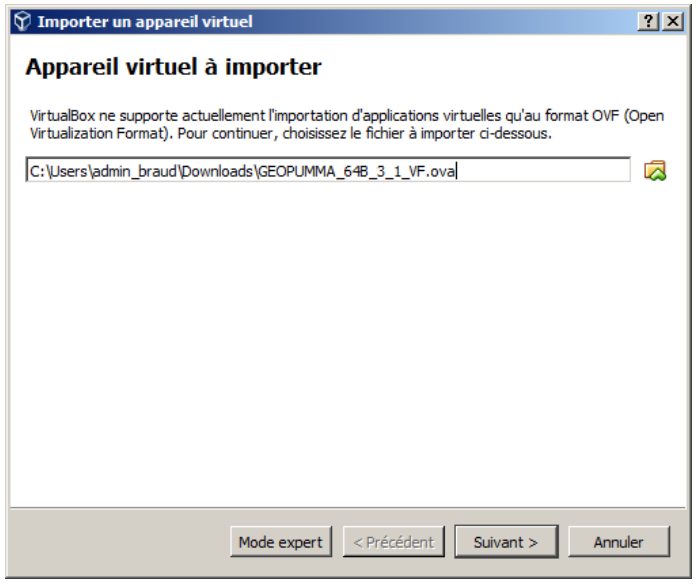
Step: Installation of the Virtual box

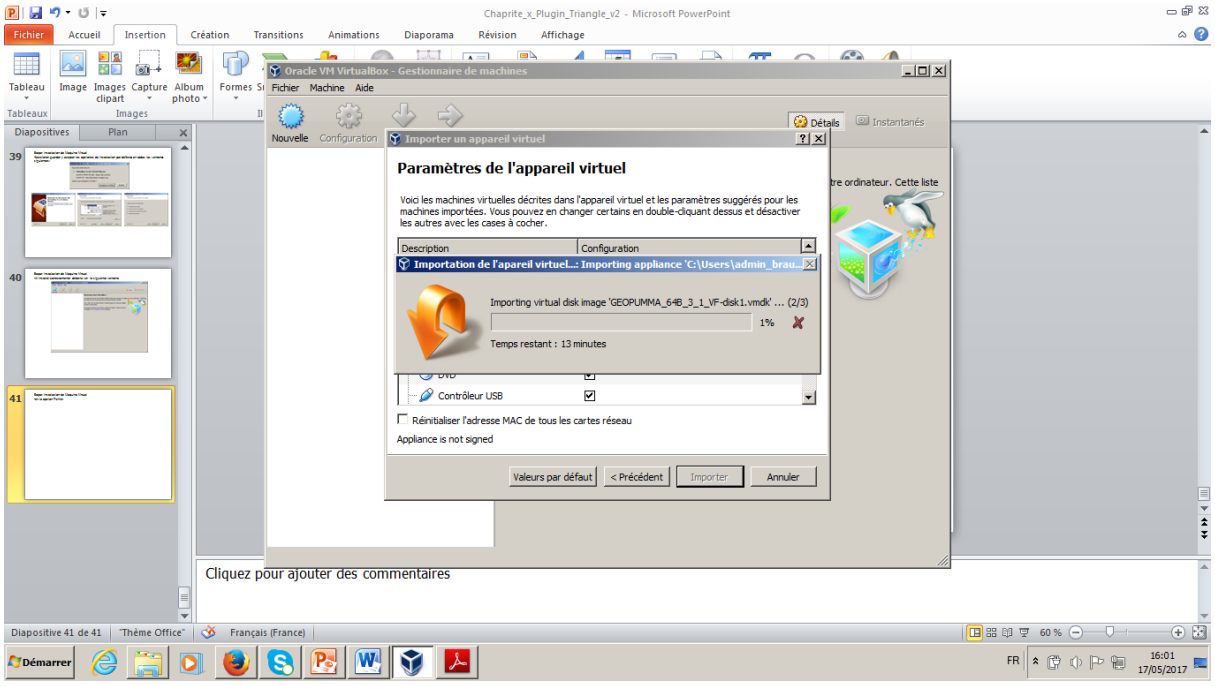
-If the installation ran correctly, the following window should appear:



Step: Installation of the Virtual box
-In the « File » option

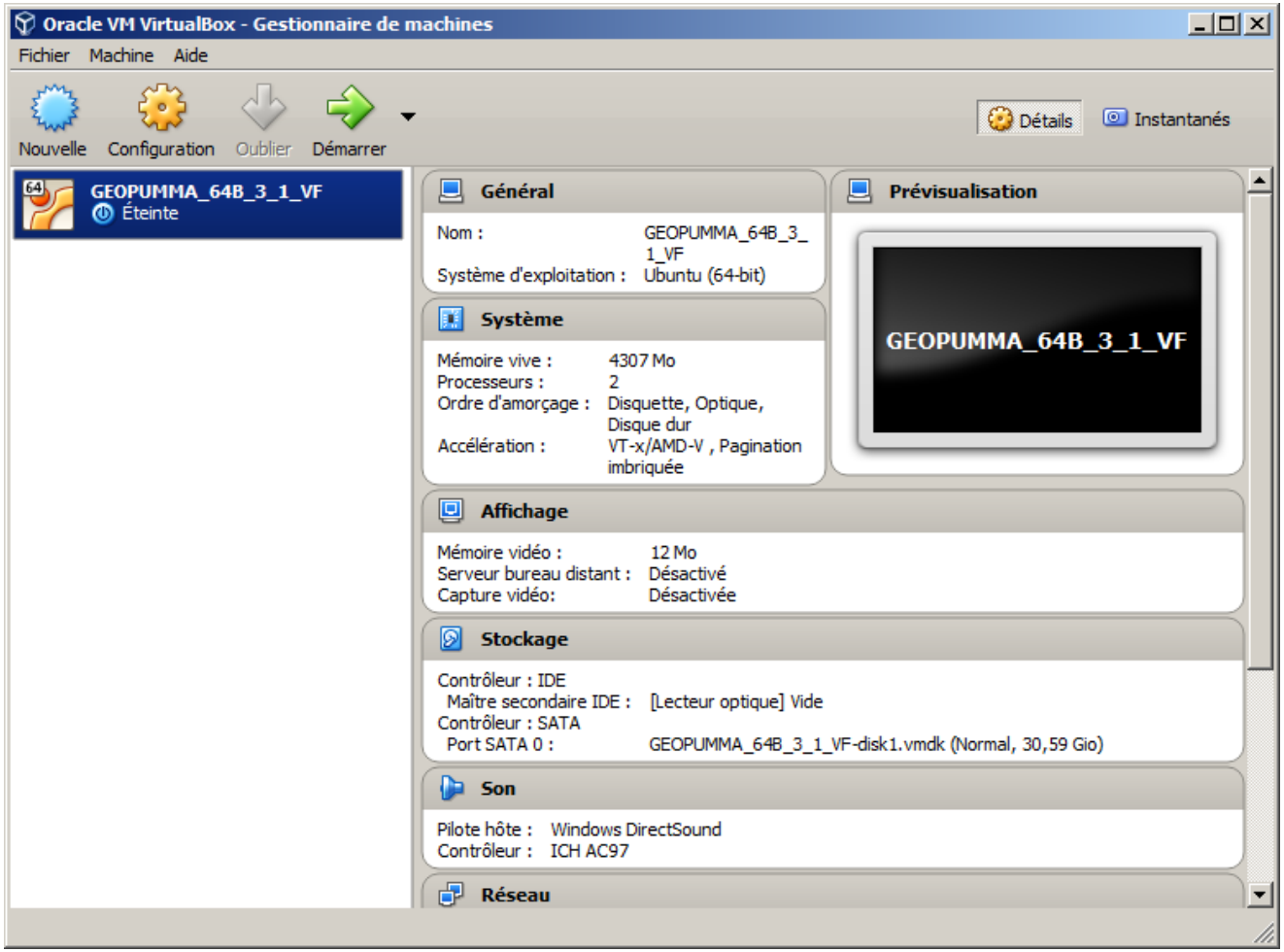






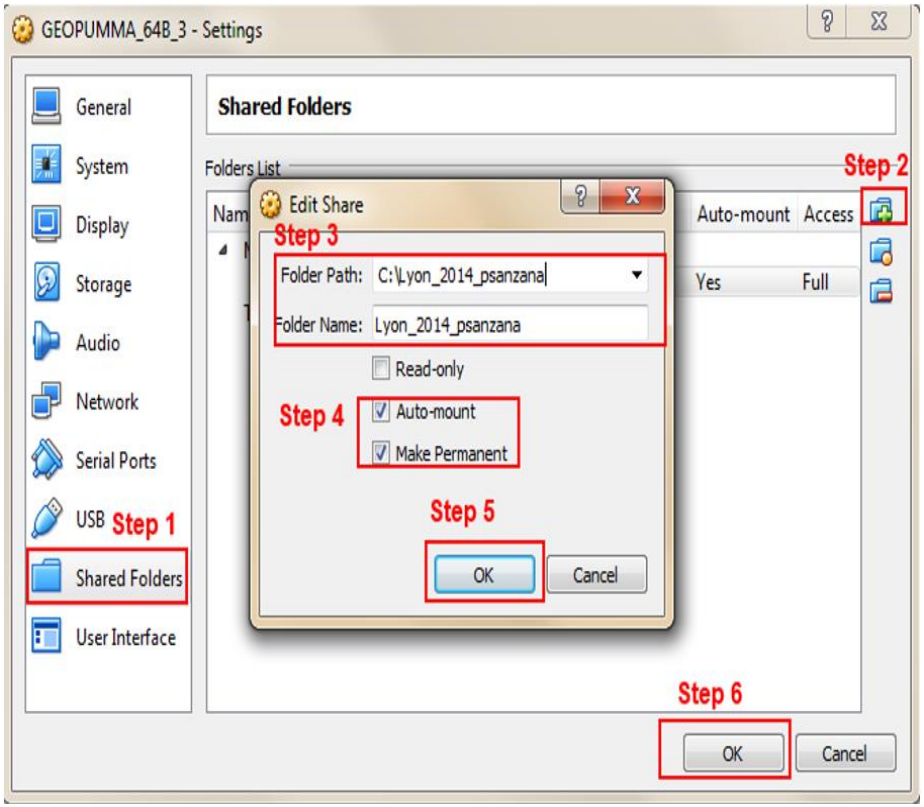
Step: Installation of the Virtual box

-Select « Initialize the Virtual box ». In case of problem in this step, it is possible to find solutions on this web page https://forge.irstea.fr/projects/geopumma/files and to download the tutorial Tutorial Geo-PUMMA v1.1.pdf



Define a shared folder that can be accessed from the Virtual Box and the main system. To achieve that:

- Go to “Shared folders” (Step 1)
- Select the wished folder (Step 2) and click the icon ”add a folder”
- Create a folder named “Lyon_2014_psanzana” in the example(Step 3)
- Select also options “Auto-mount” and “Make permanent” (Step 4).
-Then click “Ok” to close the window (Step 5)
- Then click “Ok” again to add the folder (Step 6).

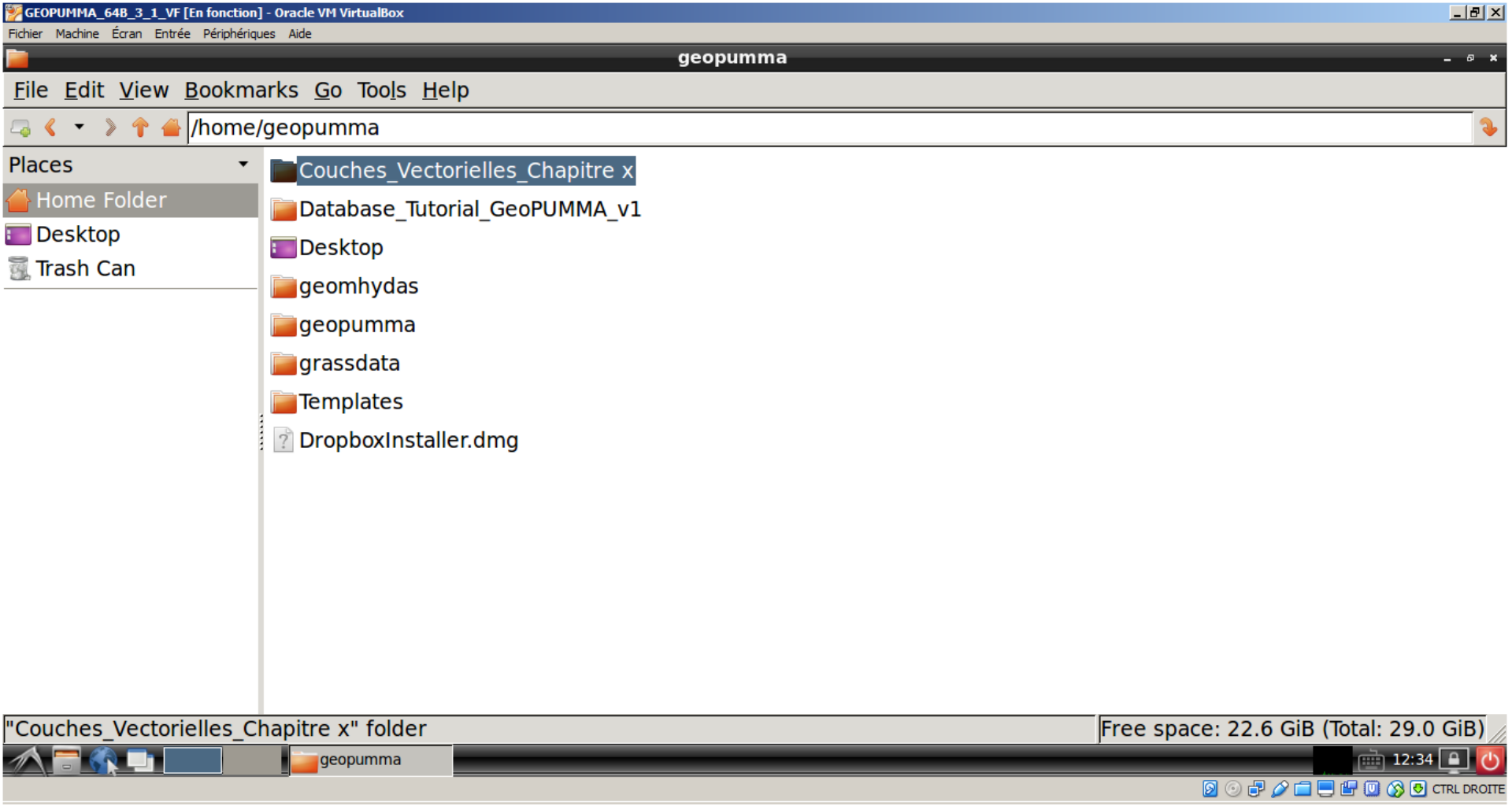


Step: Installation of the Virtual box

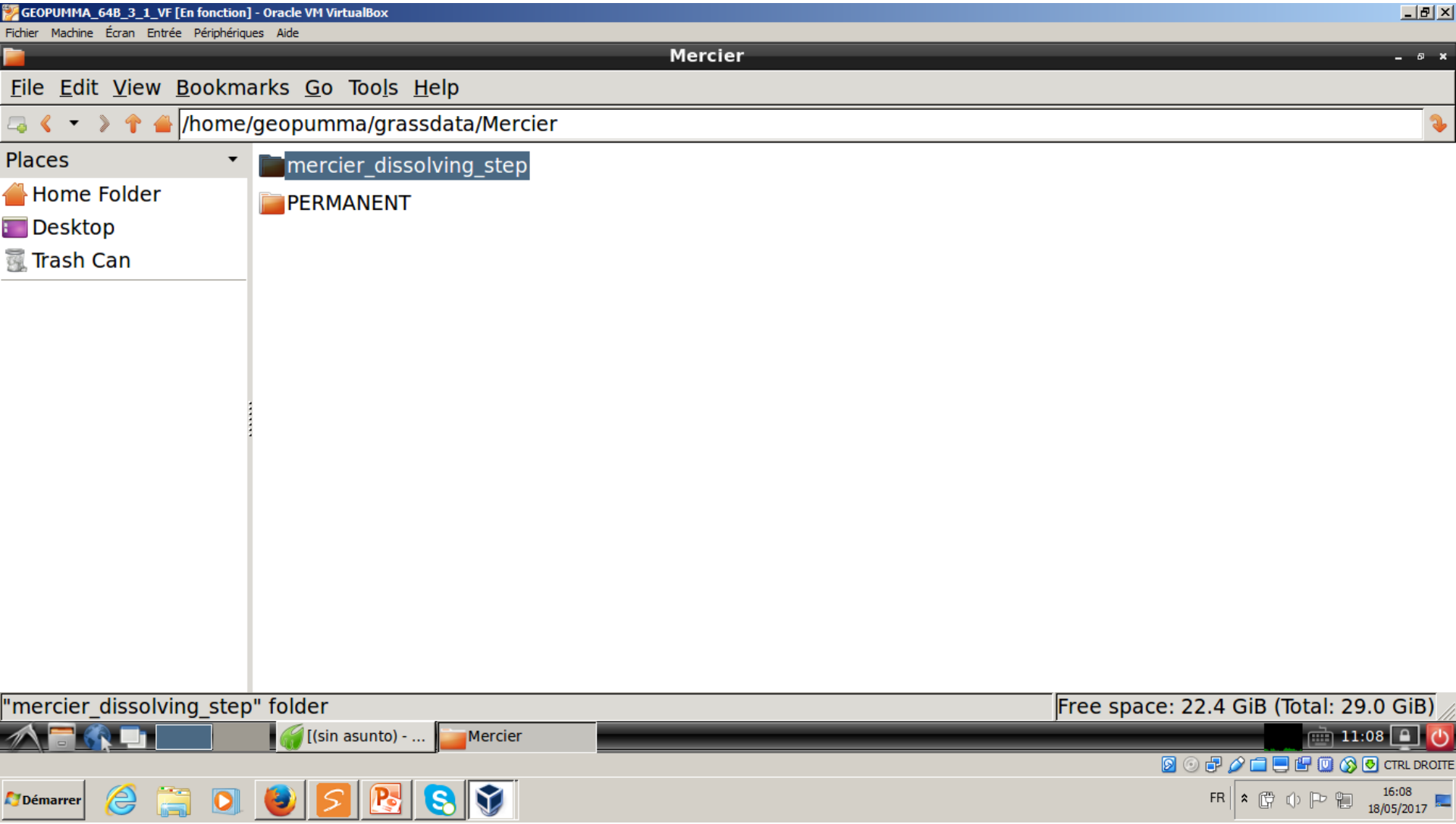
-Once the virtual box is launched, the following screen should appear:

Step: Dissolution of triangulated elements

-Copy the archive Couches_Vectorielles_Chapitre_x in the folder /home/geopumma as shown in the figure below



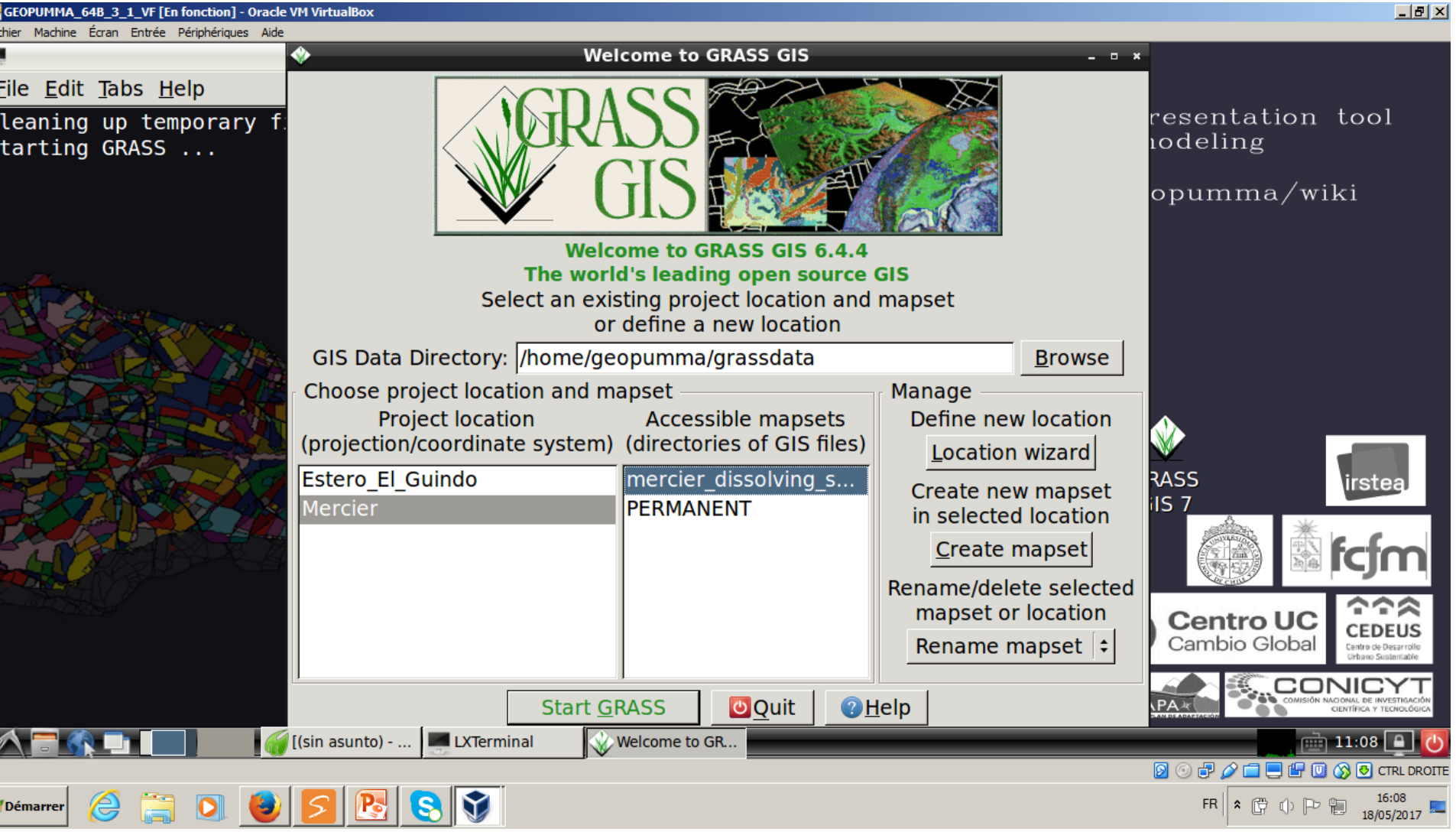
Step: Dissolution of triangulated elements
-Select the folder archive and copy it in the folder /grasssdata/Mercier as shown in the figure below



Step: Dissolution of triangulated elements

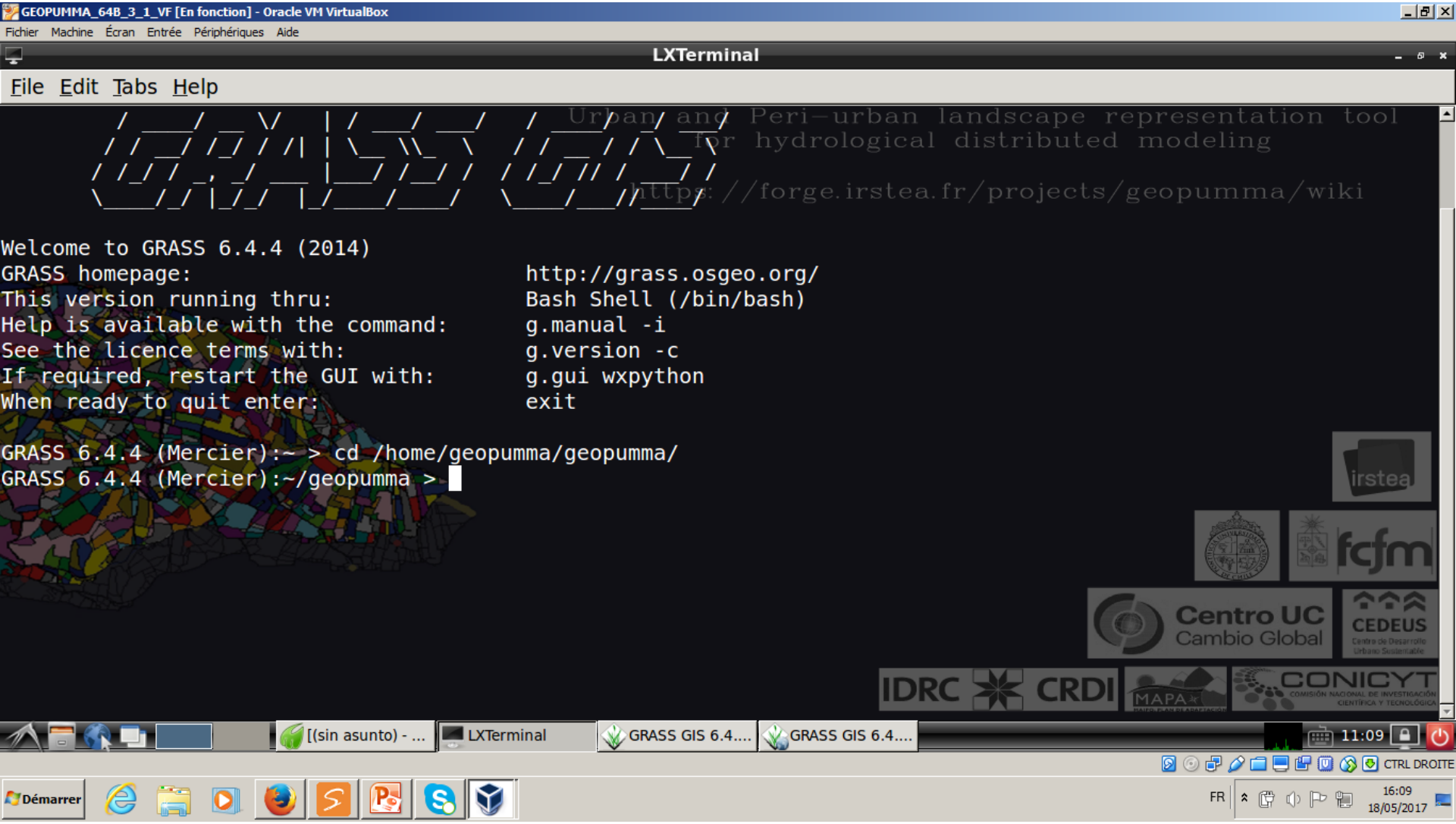
-Select the GRASS GIS icon that corresponds to the 6.4 version:

Step: Dissolution of triangulated elements
-Select the Mercier and the mercier_dissolving_step folder



Step: Dissolution of triangulated elements

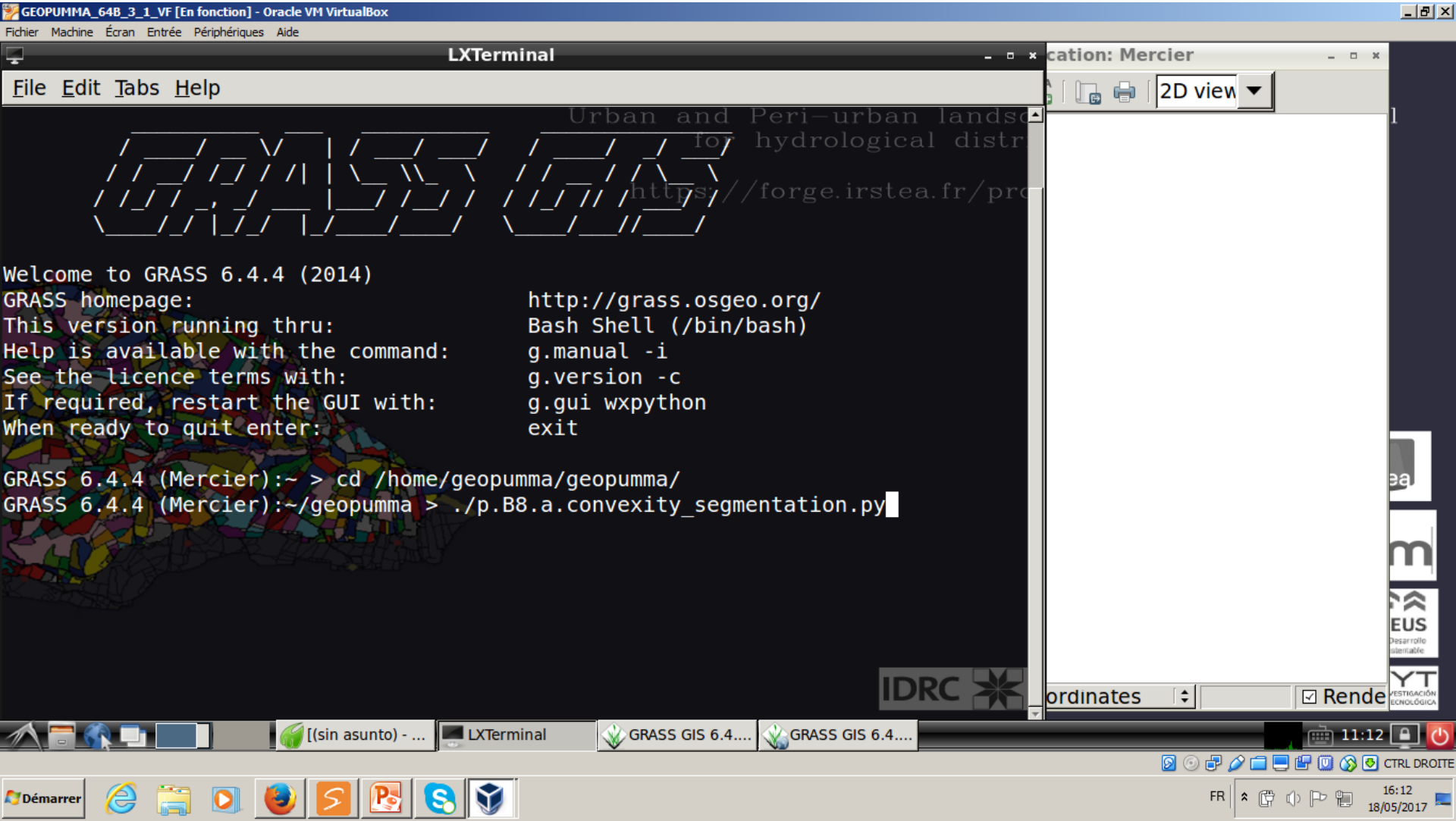
-Once GRASS is open, run the following commands



Step: Dissolution of triangulated elements

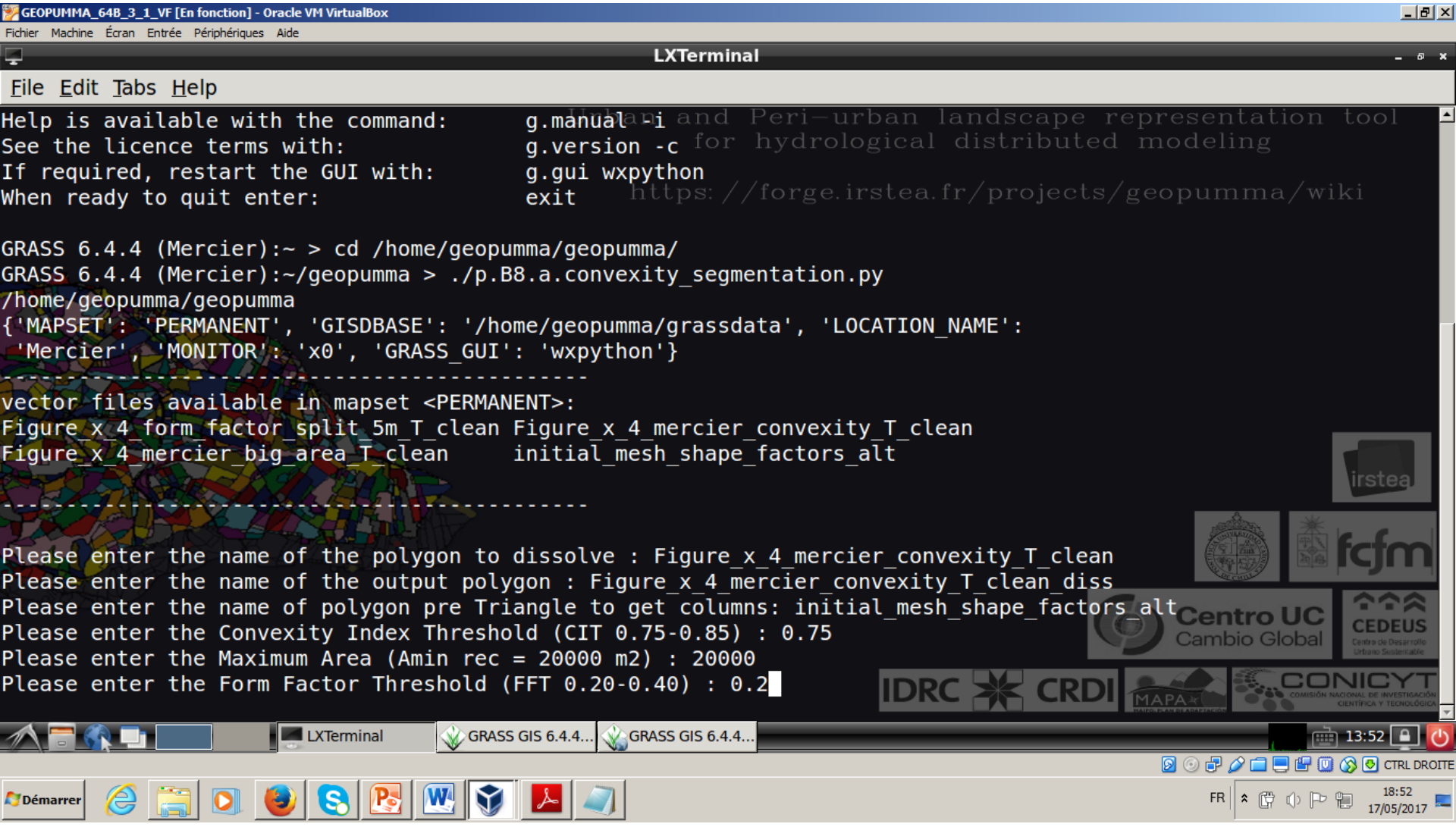
-Type cd /home/geopumma/geopumma/ in the command line

-Type ./p.B8.a.convexity_segmentation.py in the command line



Step: Dissolution of triangulated elements

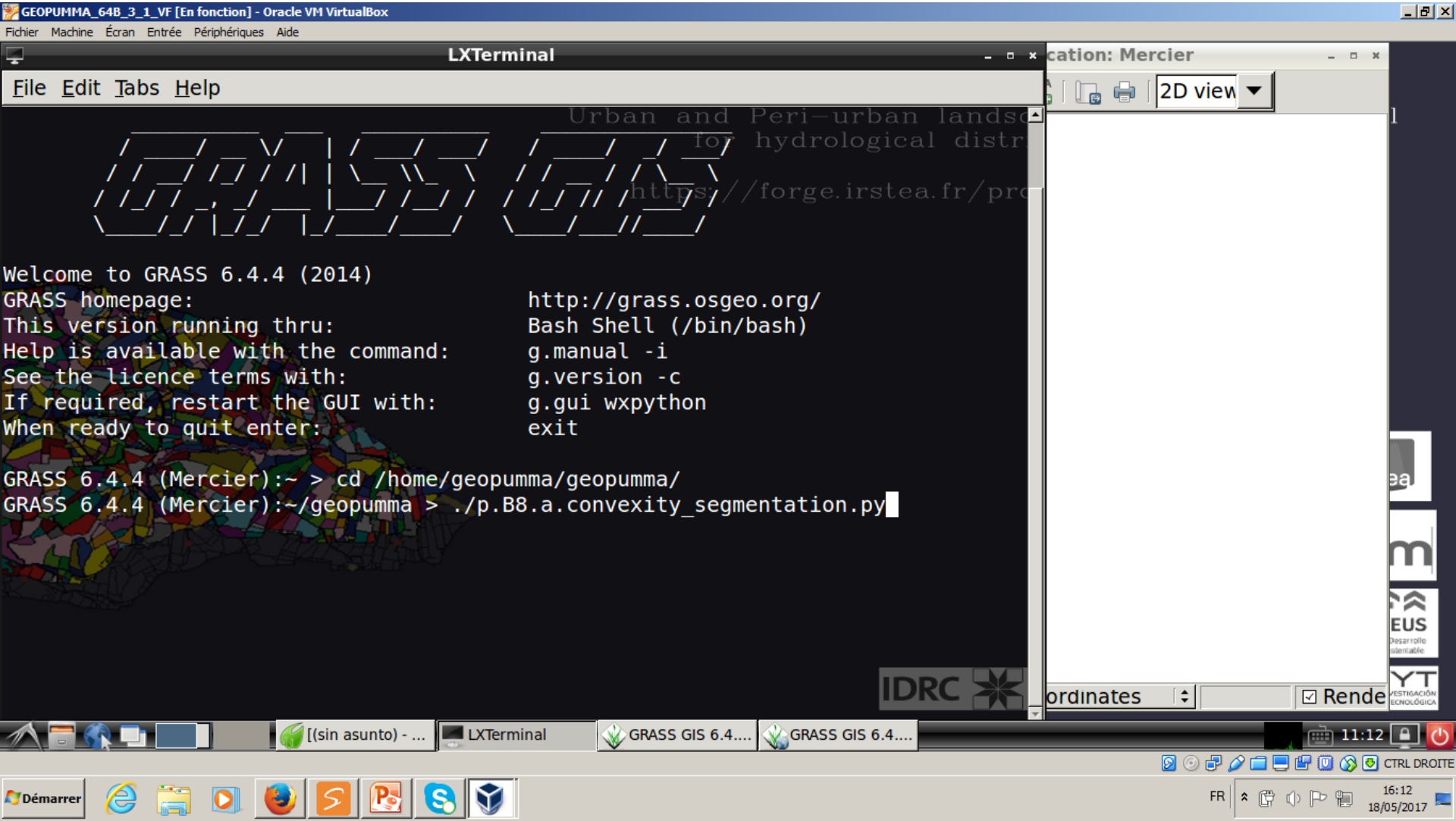
-Select the following options for the polygons selected according to the convexity criterion



Step: Dissolution of triangulated elements

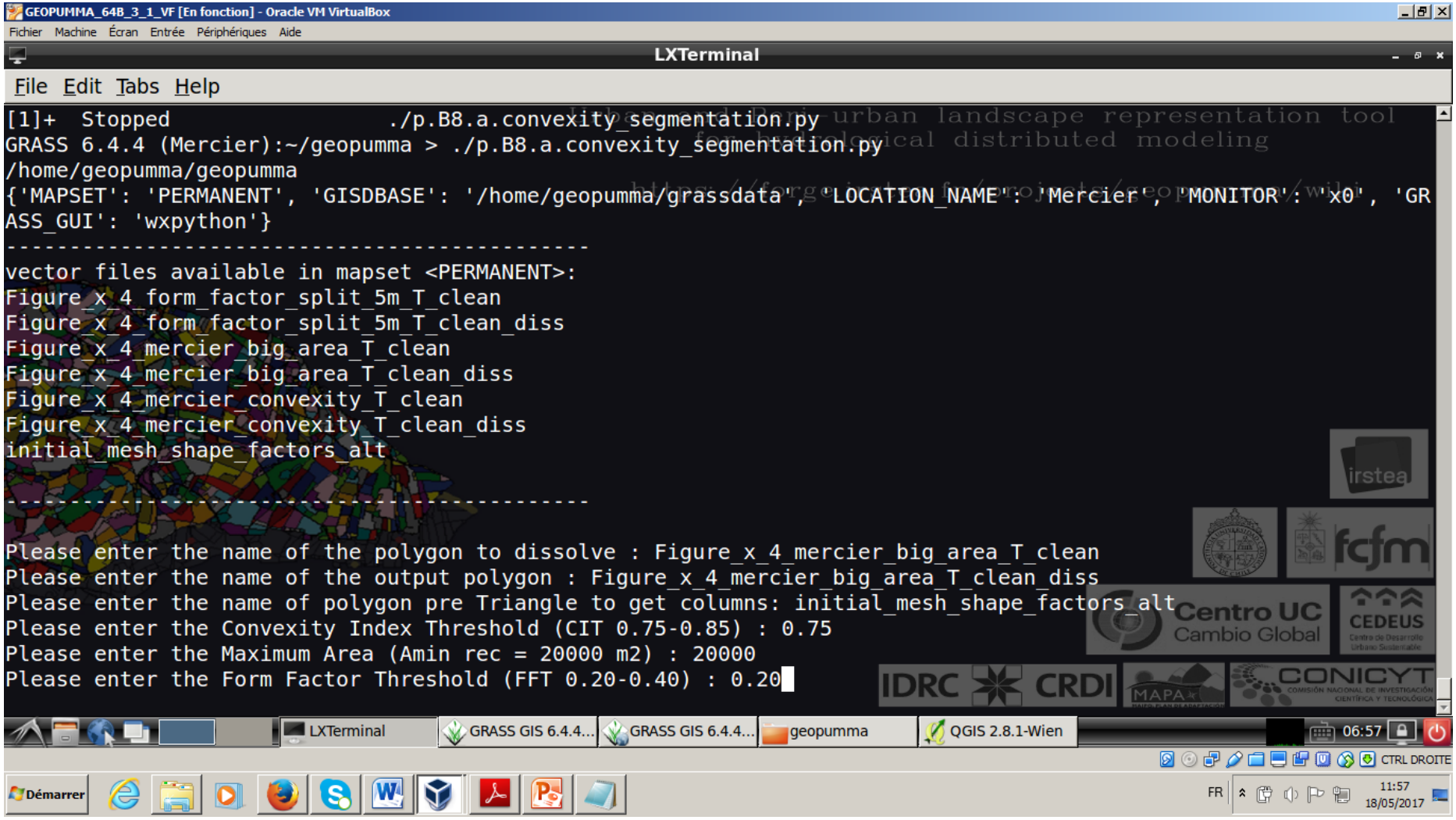
-Type cd /home/geopumma/geopumma/ in the command line

-Type ./p.B8.a.convexity_segmentation.py in the command line



Step: Dissolution of triangulated elements

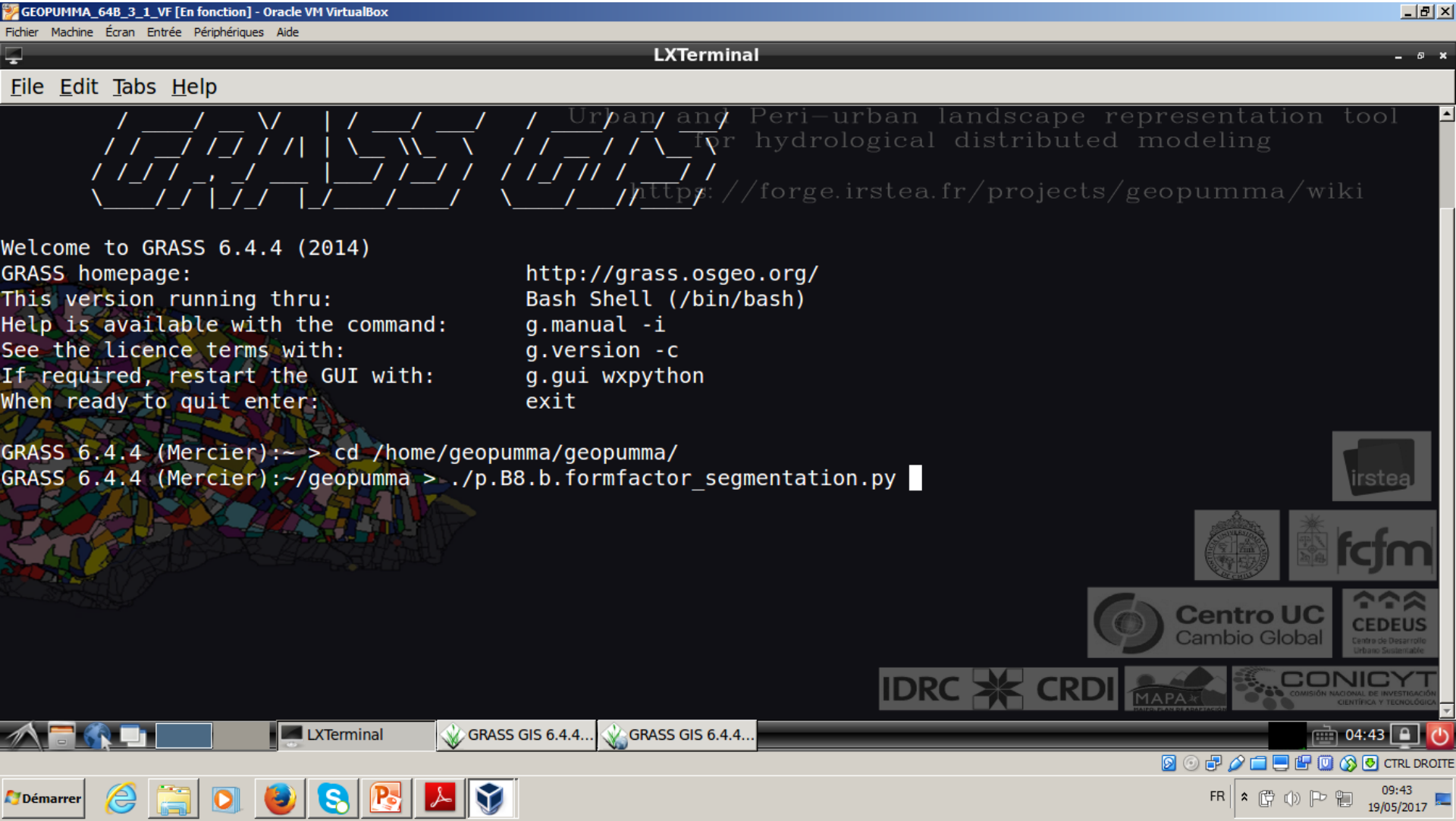
-Select the following options for the polygons selected according to the area criterion



Step: Dissolution of triangulated elements

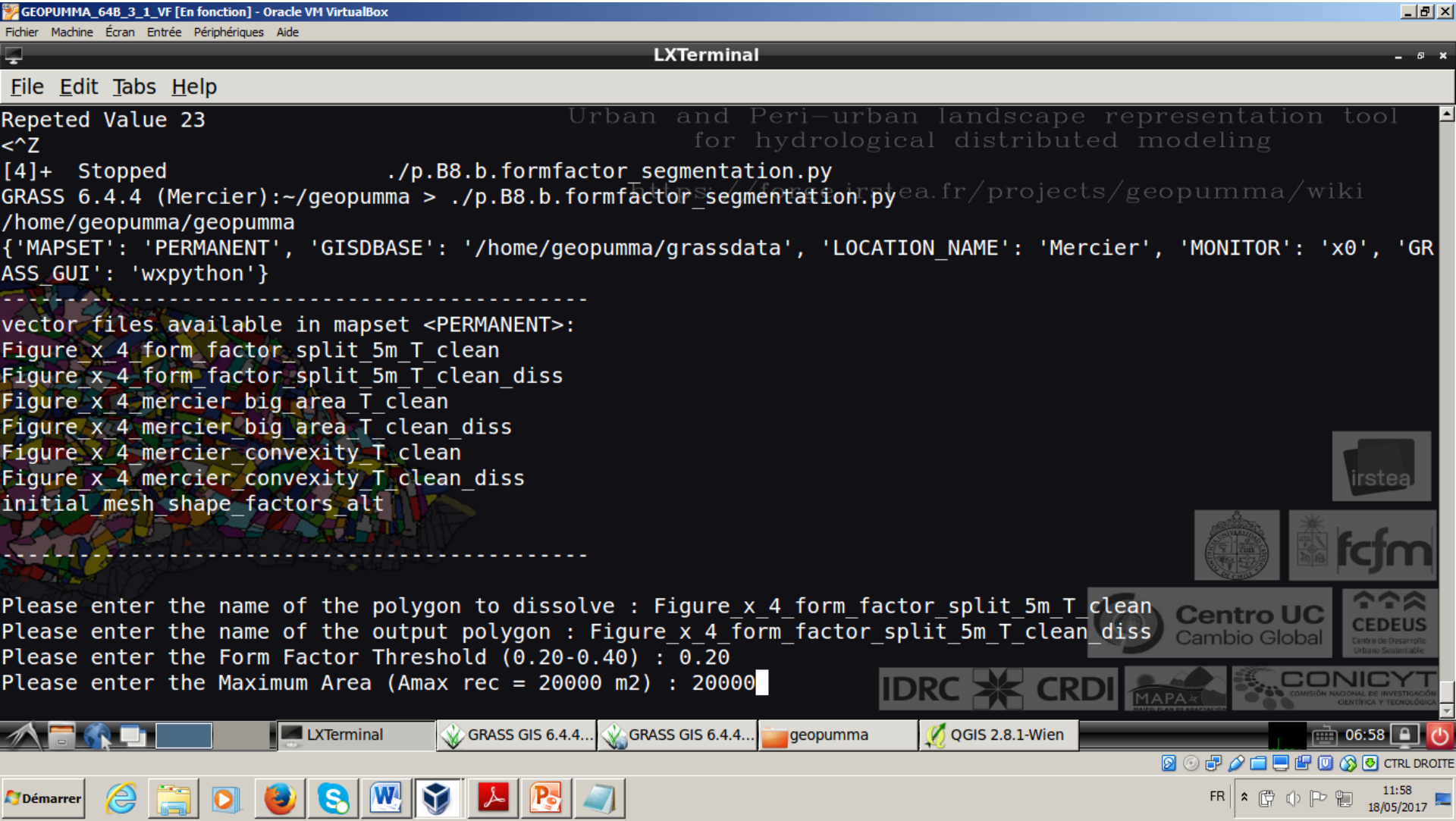
-Type cd /home/geopumma/geopumma/ in the command line

-Type ./p.B8.b.formfactor_segmentation.py in the command line

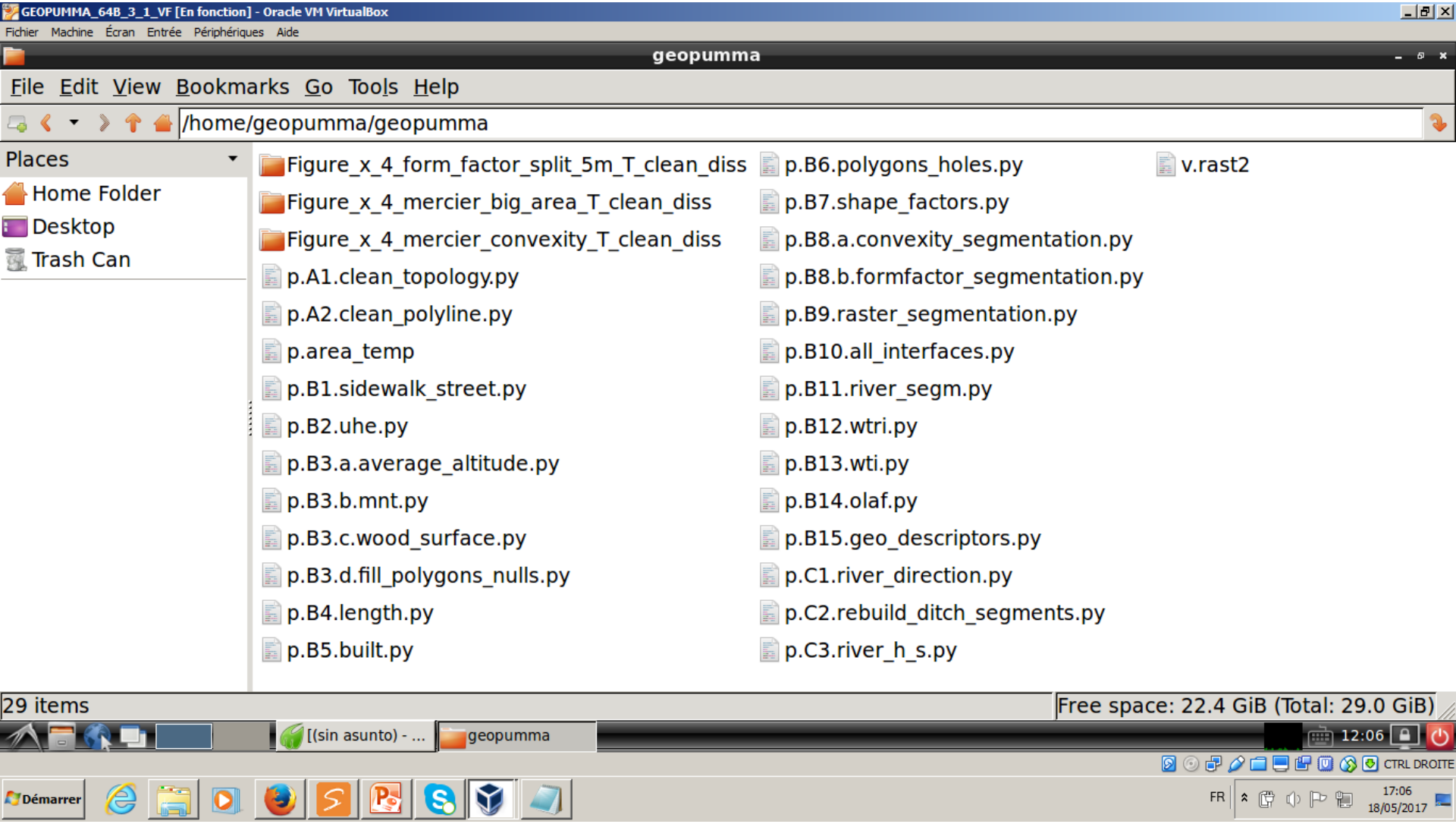


Etape: Dissolution d'éléments triangulés

-Select the following options for the polygons selected according to the form factor criterion

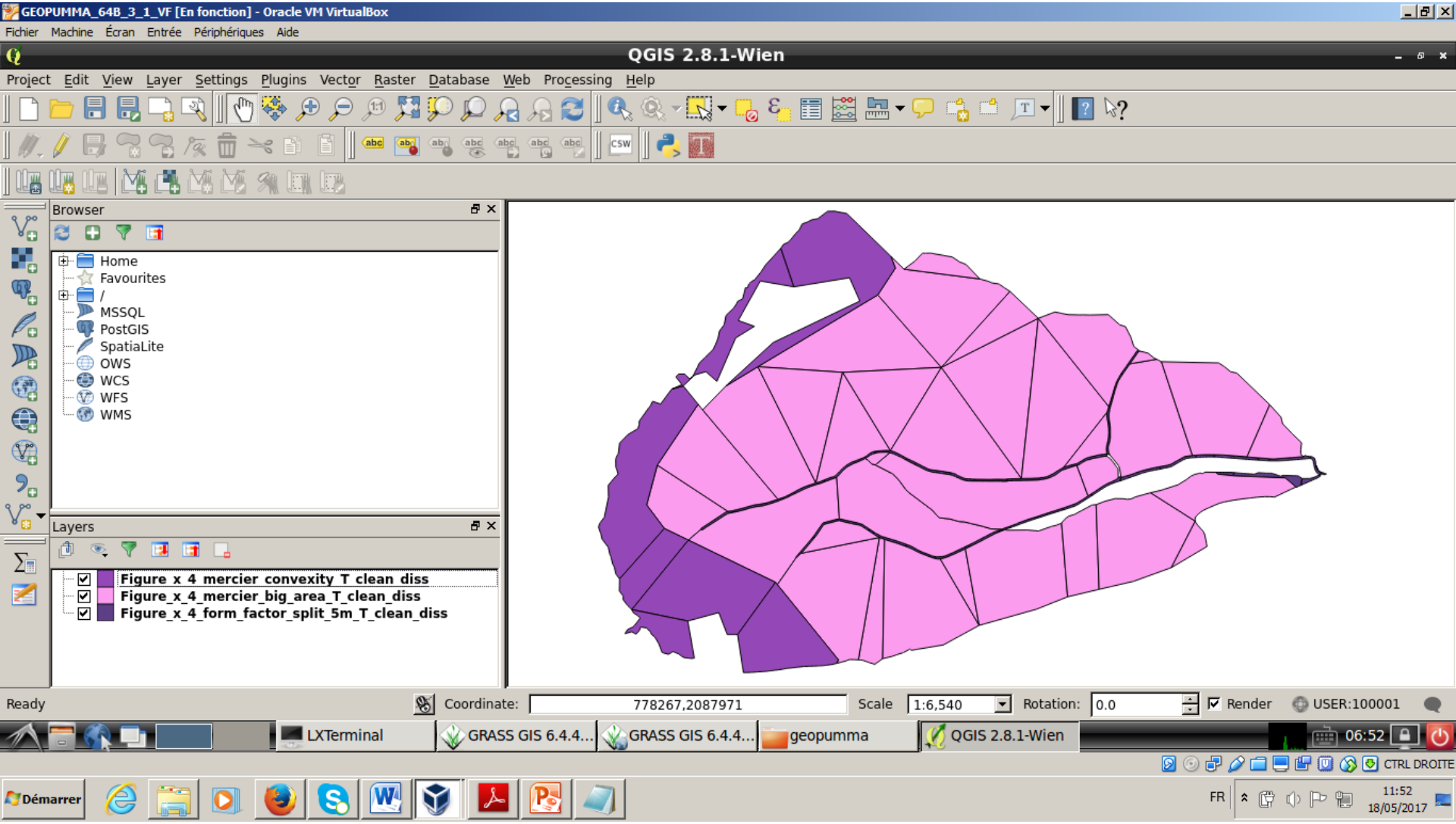


Step: Dissolution of triangulated elements
-The result leads to the following folders



Step: Dissolution of triangulated elements

-Each folder contains the following .shp layers (Figures x.34 x.35 y x.36)



With these .shape files, it is possible to get the mesh of Figure x.37:

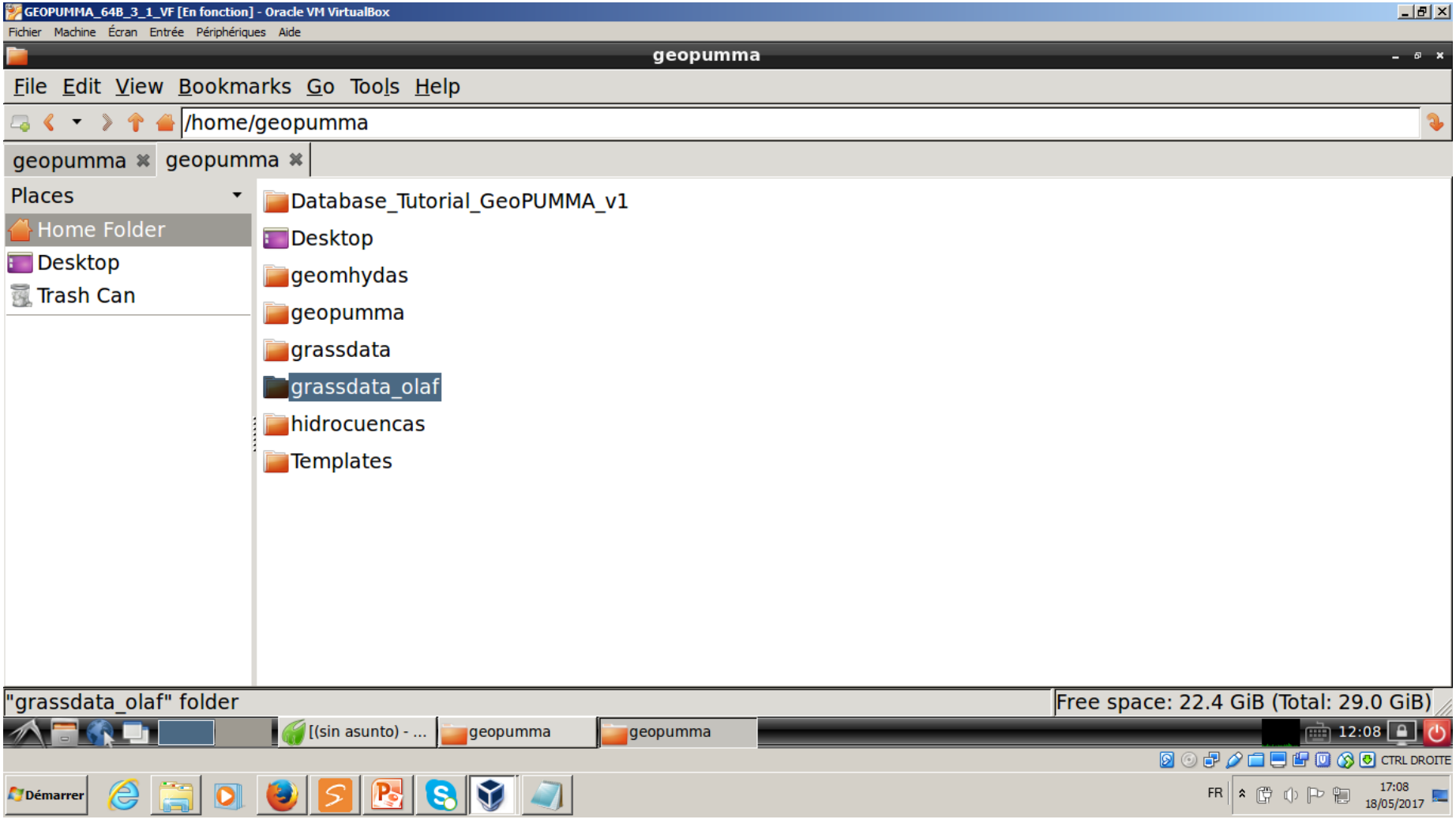
- Improved mesh on convexity and area criteria
- Improved mesh on form factor criterion

Step: Détermination of flow directions between elements (routing)

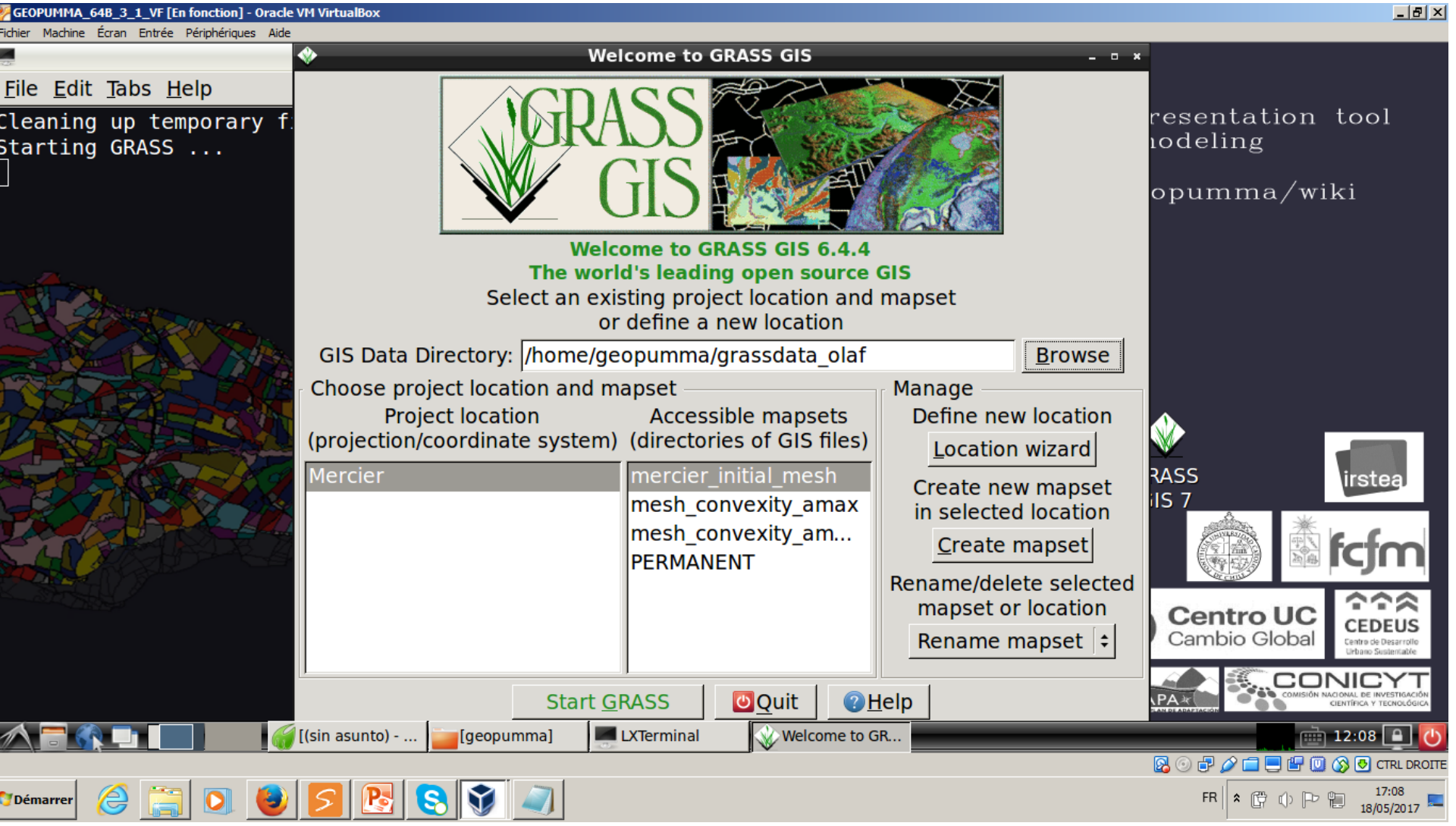
A folder with the three mesh model mesh has been prepared. The following steps only aim at illustrating how to perform the flow routing. We do not show how to get all the layers (other commands from Geo-PUMMA should be used for that), but we provide the resulting layers which are outputs of these scripts

For more details about intermediary steps, the user can read the GeoPUMMA V1,1 users' manual

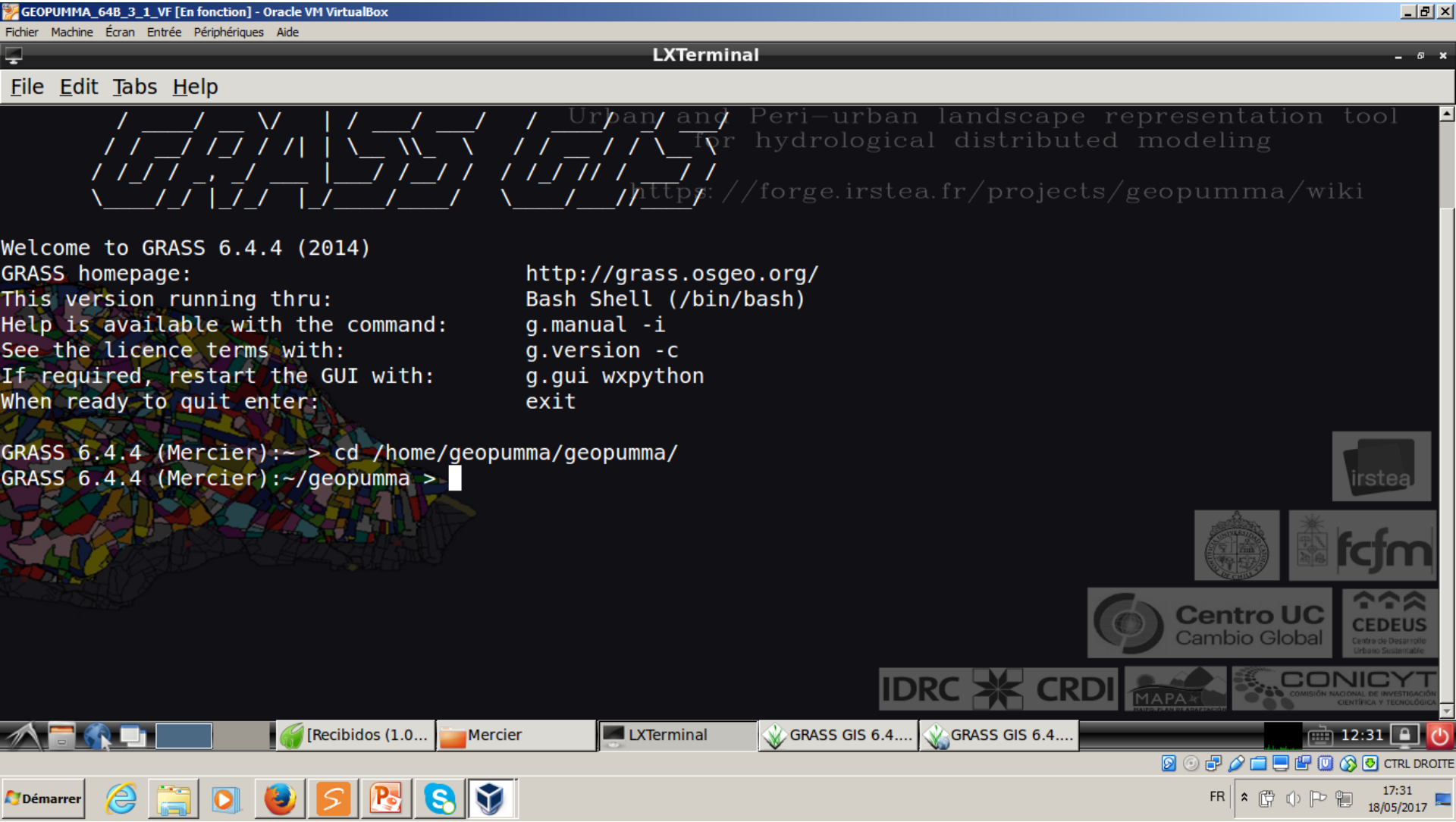
-Copy the folder grassdata_olaf in the folder: /home/geopumma



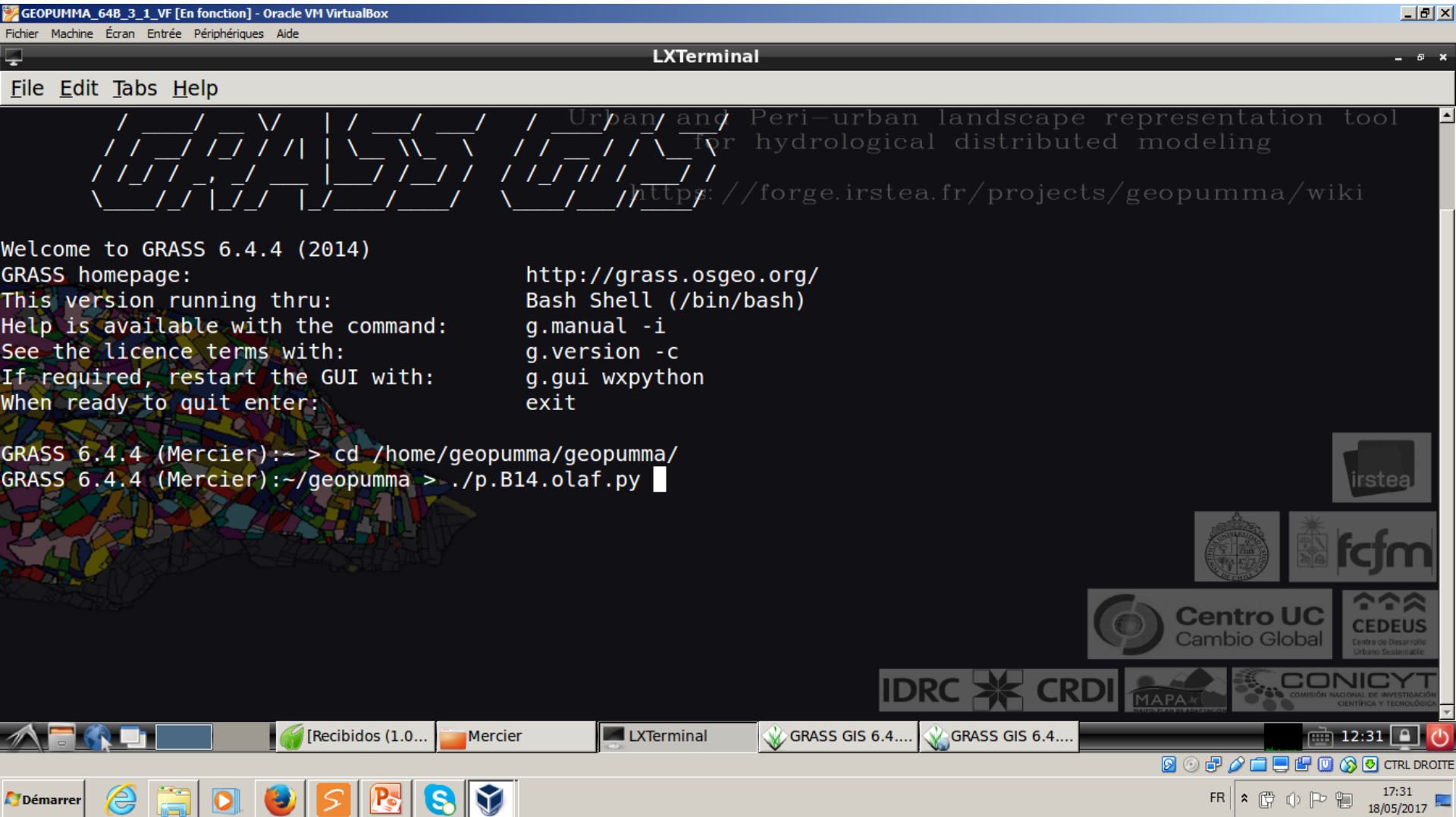
Step: Flow routing of the initial mesh
Select LOCATION: Mercier and MAPSET: mercier_initial_mesh



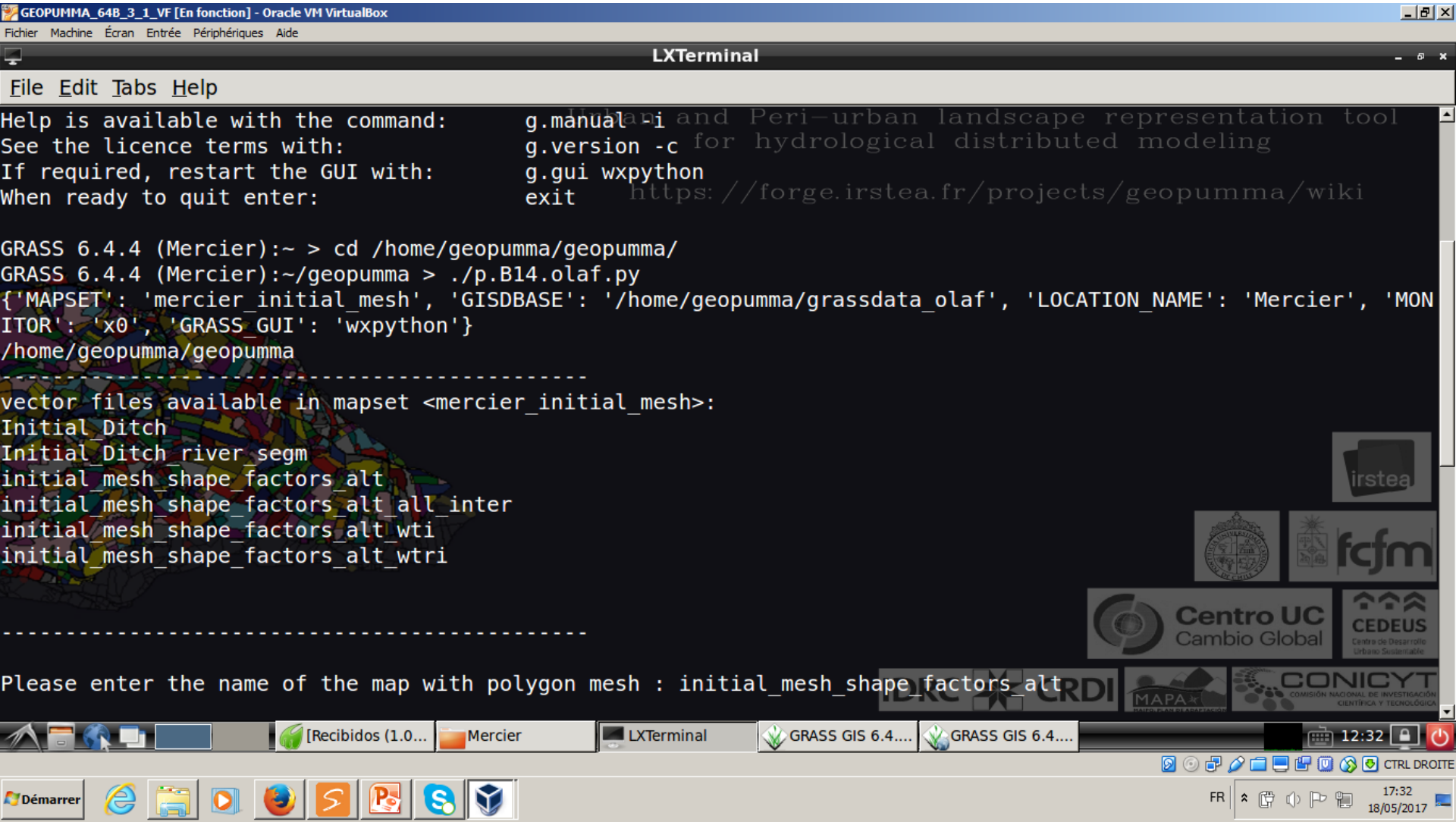
Step: Flow routing of the initial mesh
Change directory



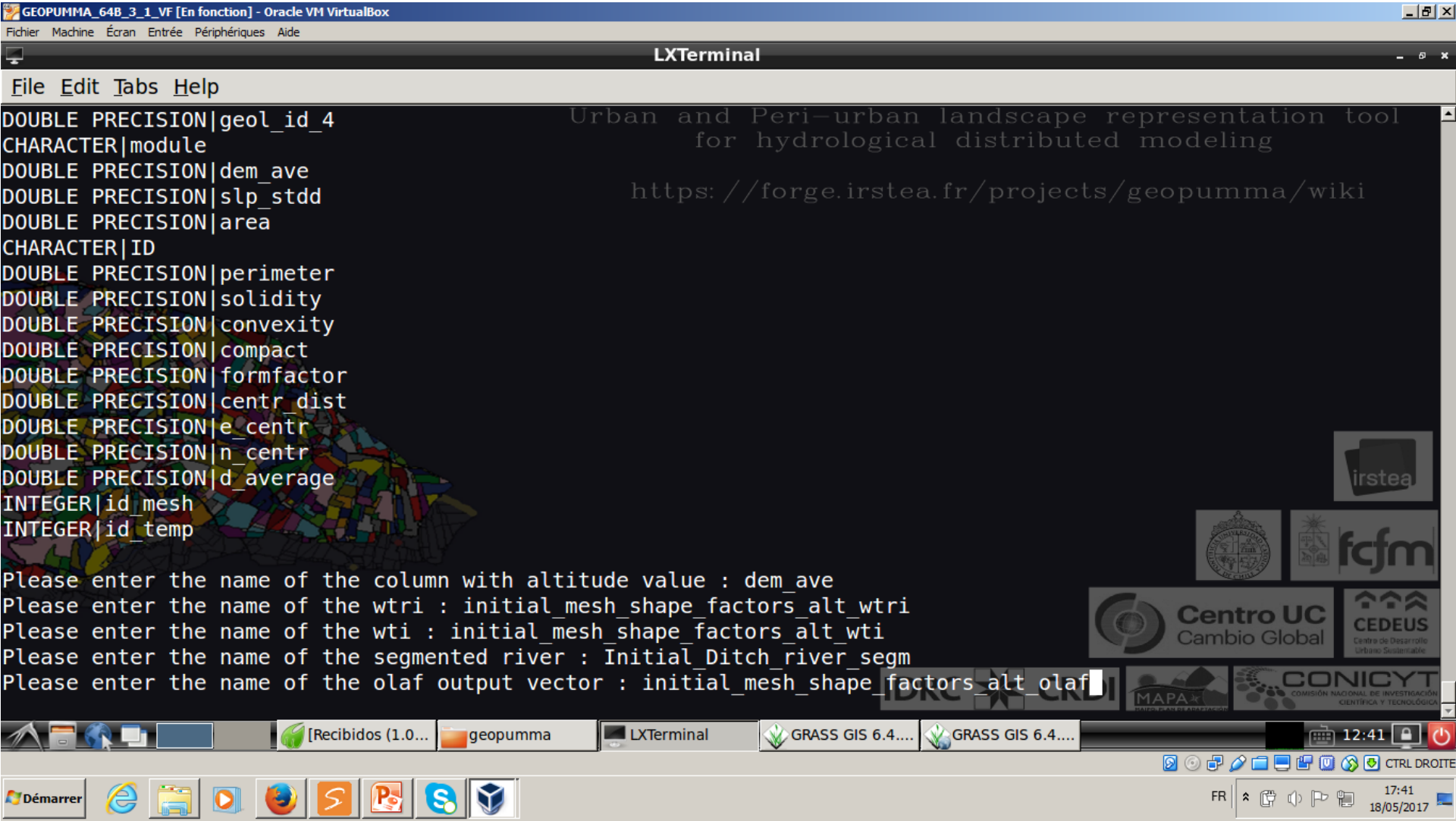
Step: Flow routing of the initial mesh
Run the olaf command



Step: Flow routing of the initial mesh
Enter the following options



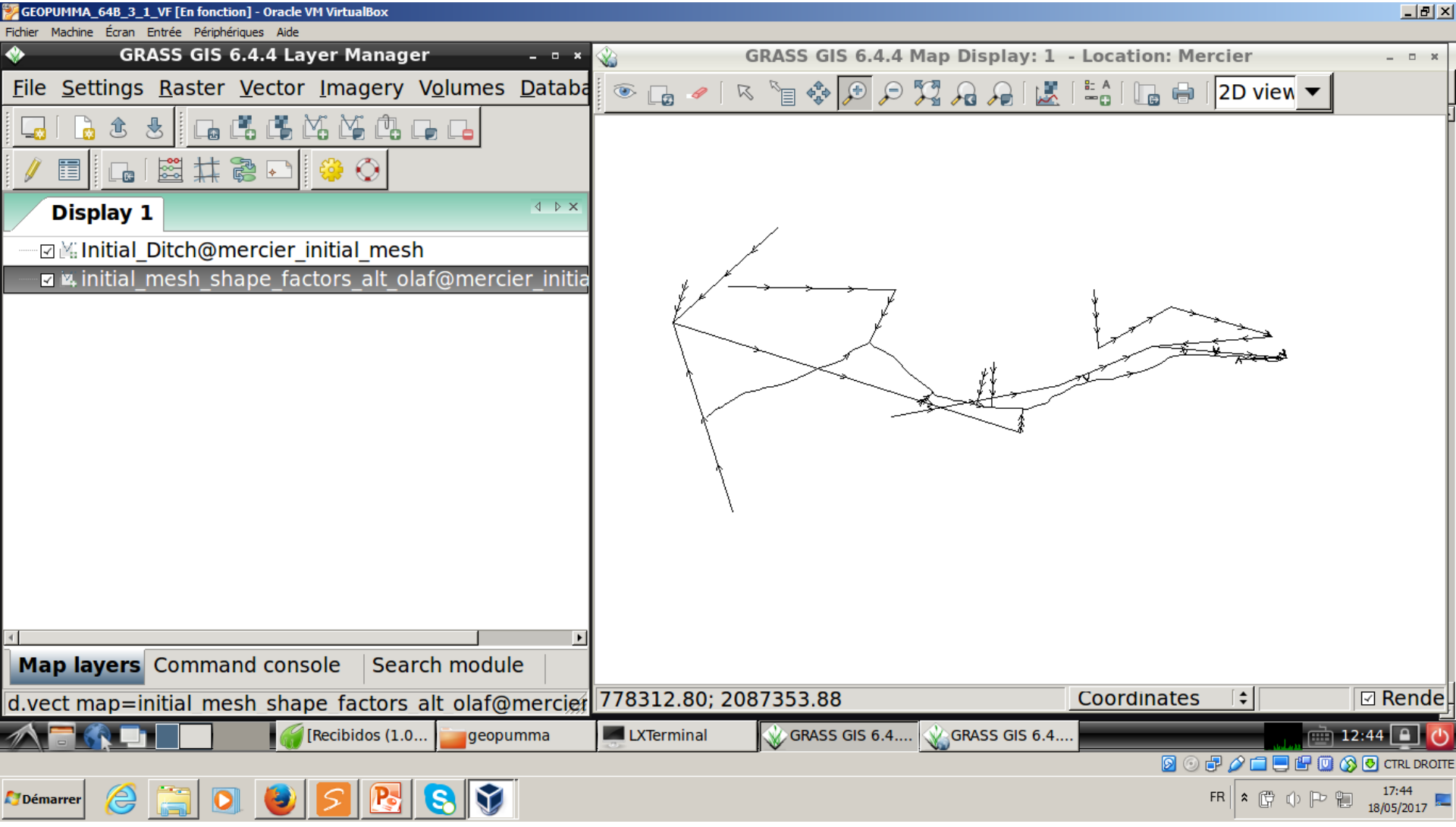
Step: Flow routing of the initial mesh
Enter the following options



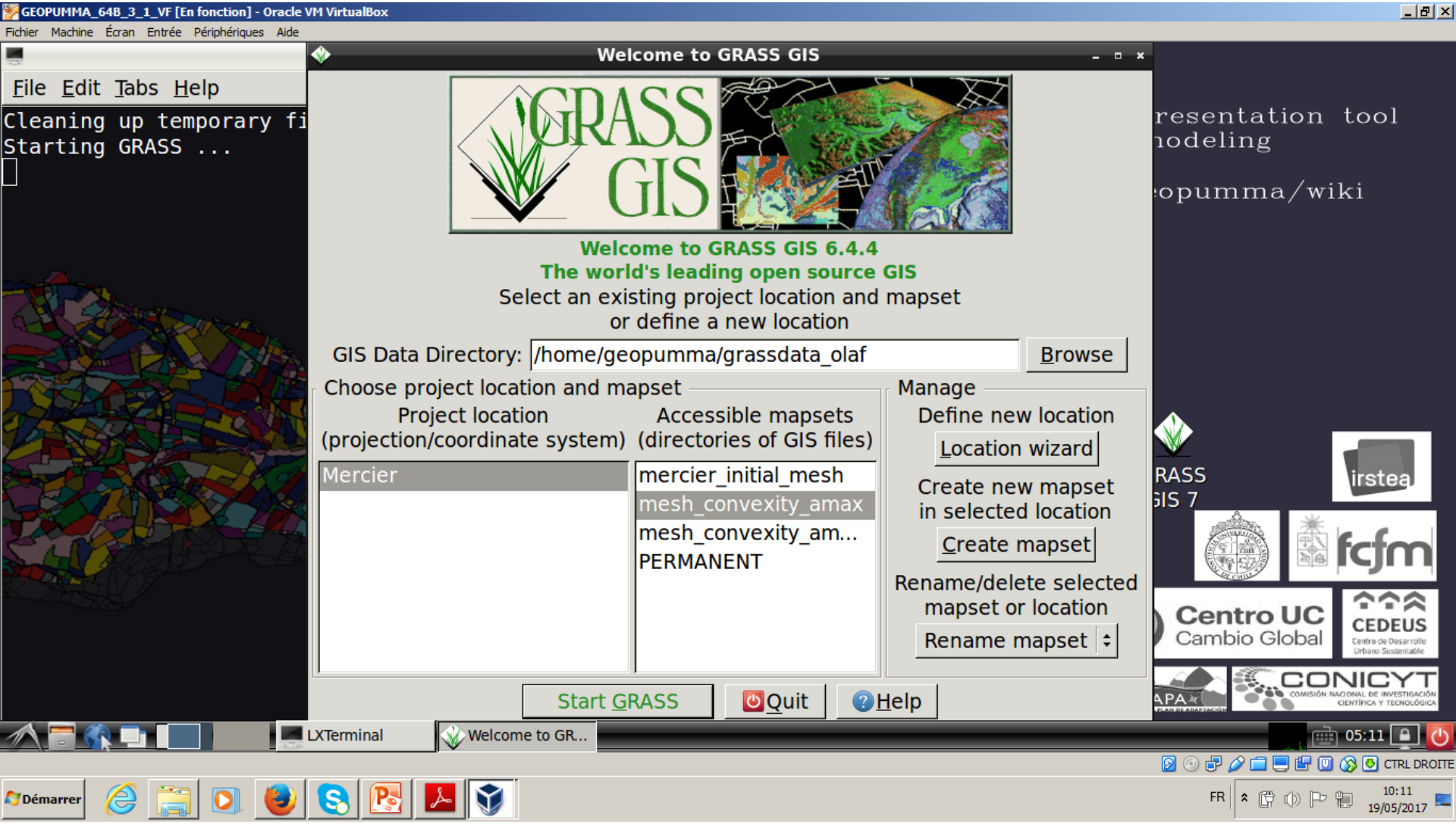
Step: Flow routing of the initial mesh

Visualize the vectorial layers corresponding to the surface drainage network in GRASS

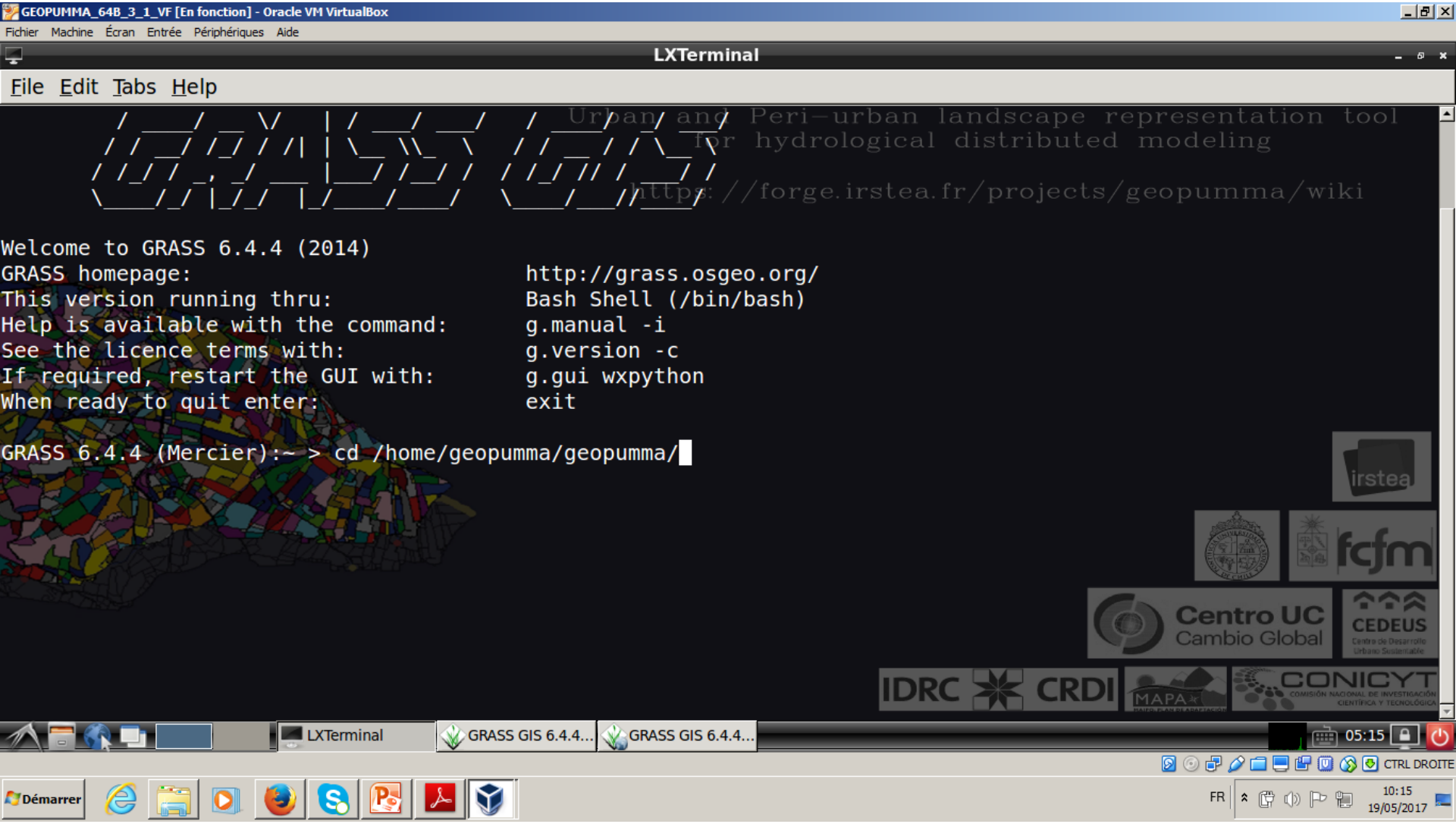
These elements belong to the drainage network shown in Figure x.37.b



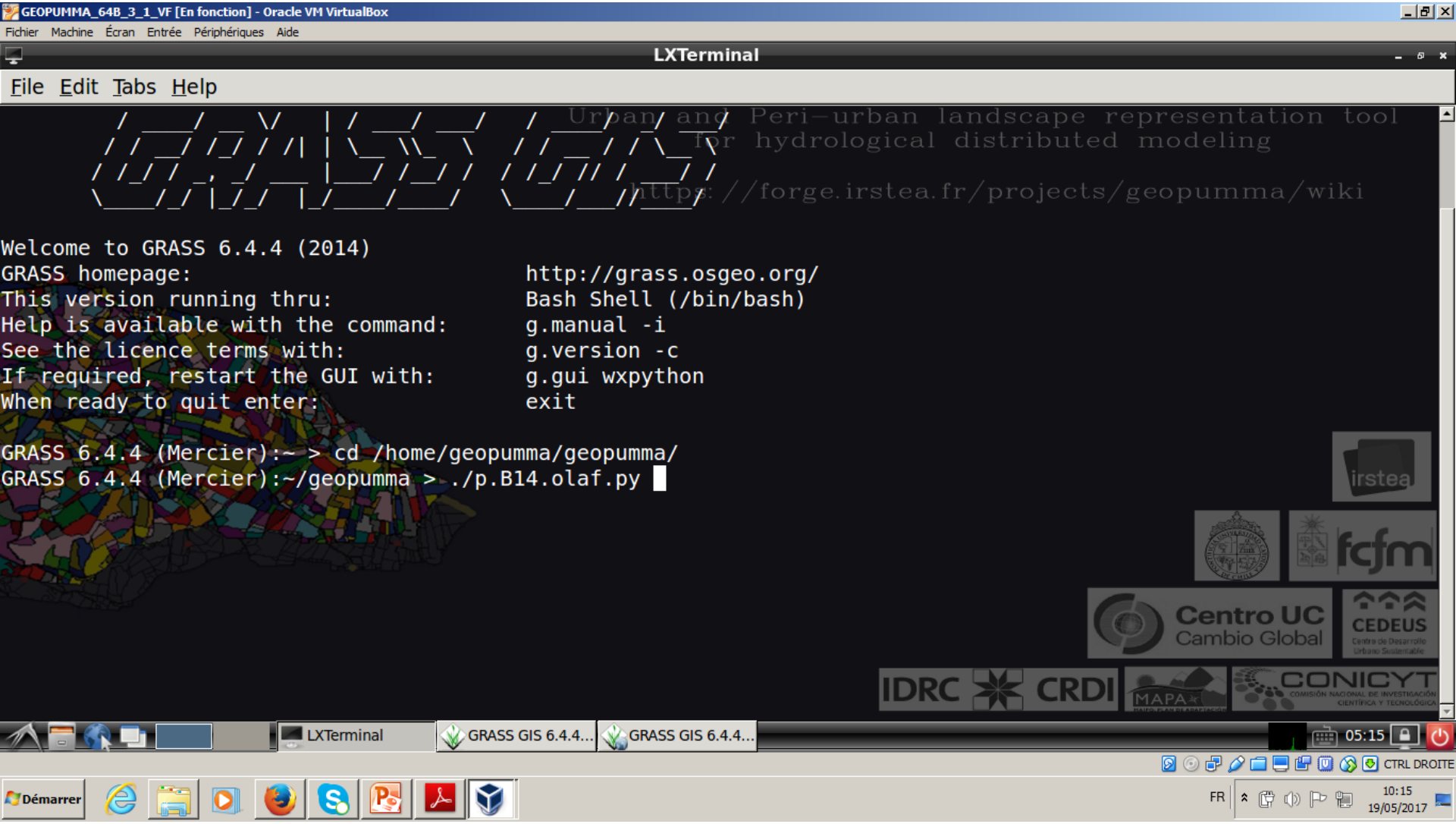
Step: Flow routing of the initial mesh
Select LOCATION: Mercier and MAPSET: mercier_convexity_amax



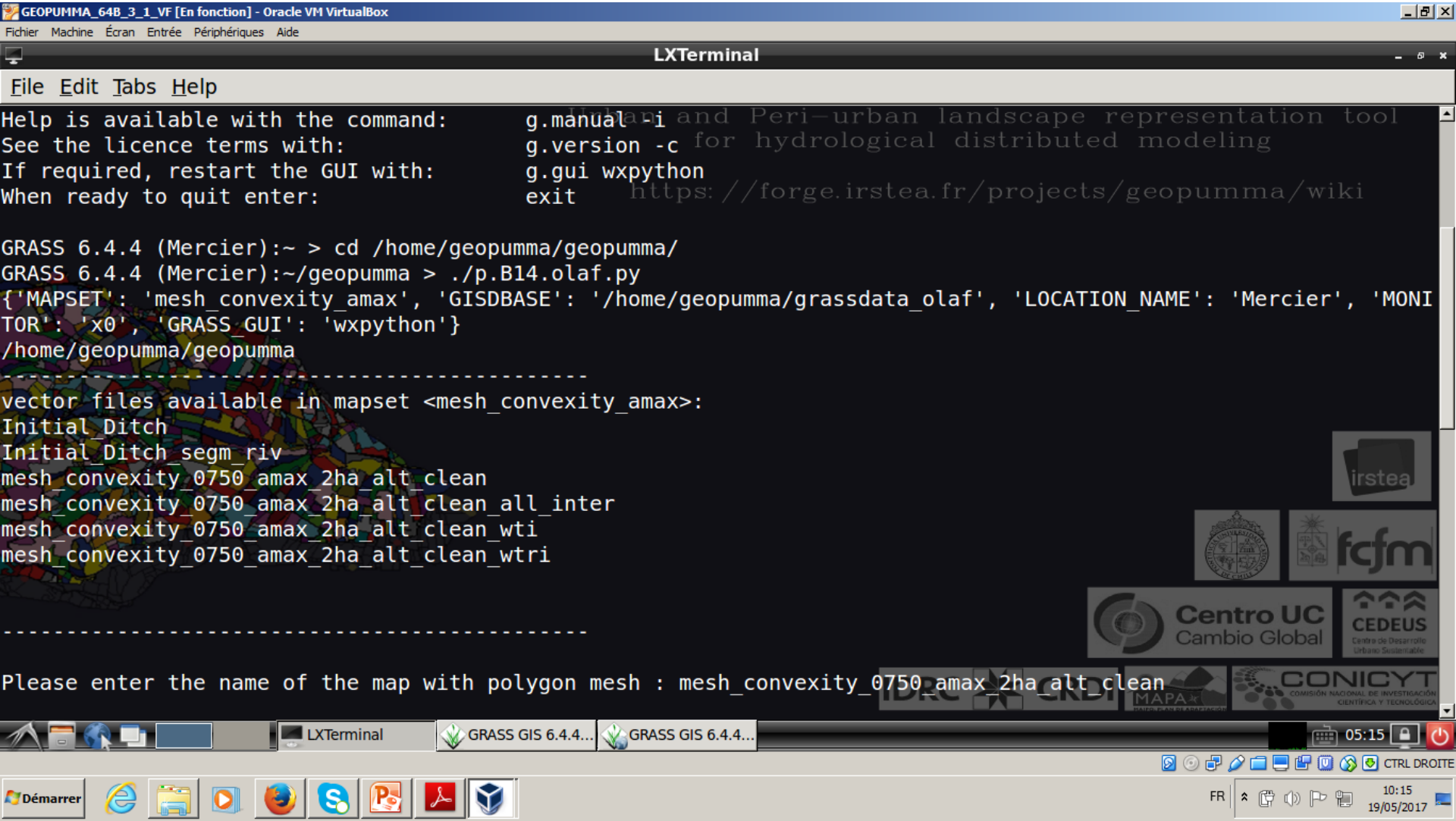
Step: Flow routing of the initial mesh
Change directory



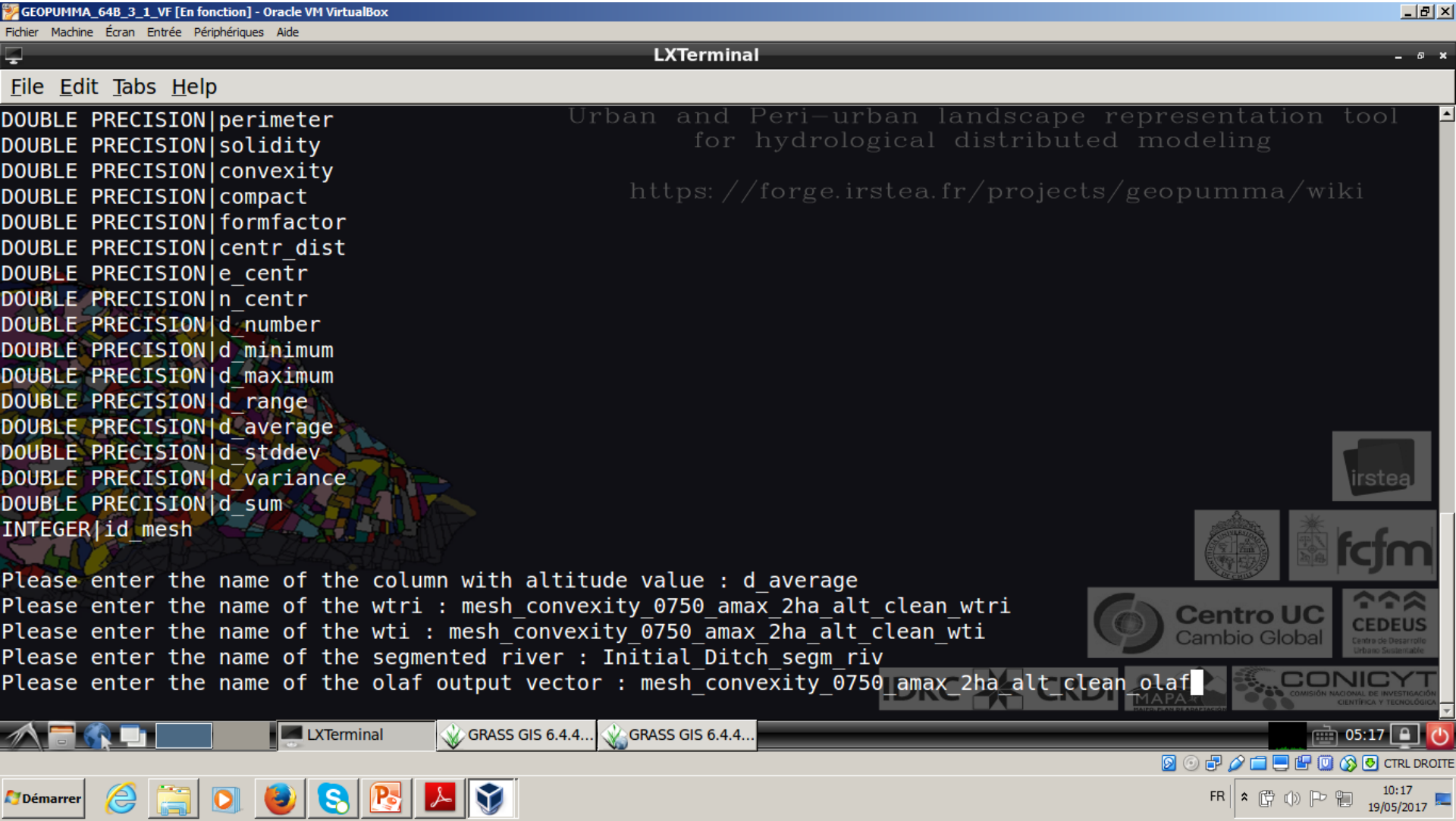
Step: Flow routing of the initial mesh
Execute the olaf command



Step: Flow routing of the initial mesh
Enter the following commands



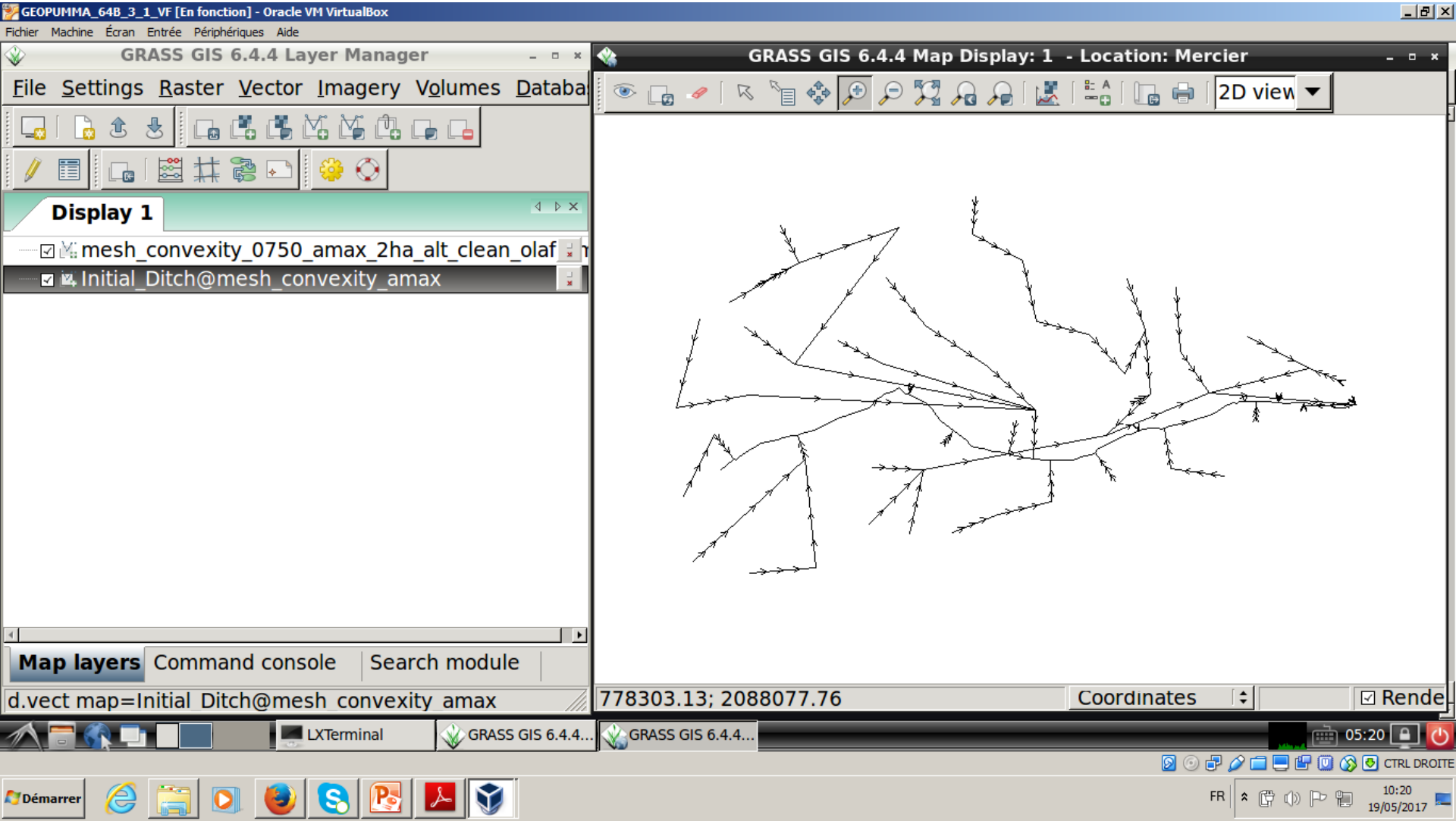
Step: Flow routing of the initial mesh
Enter the following commands



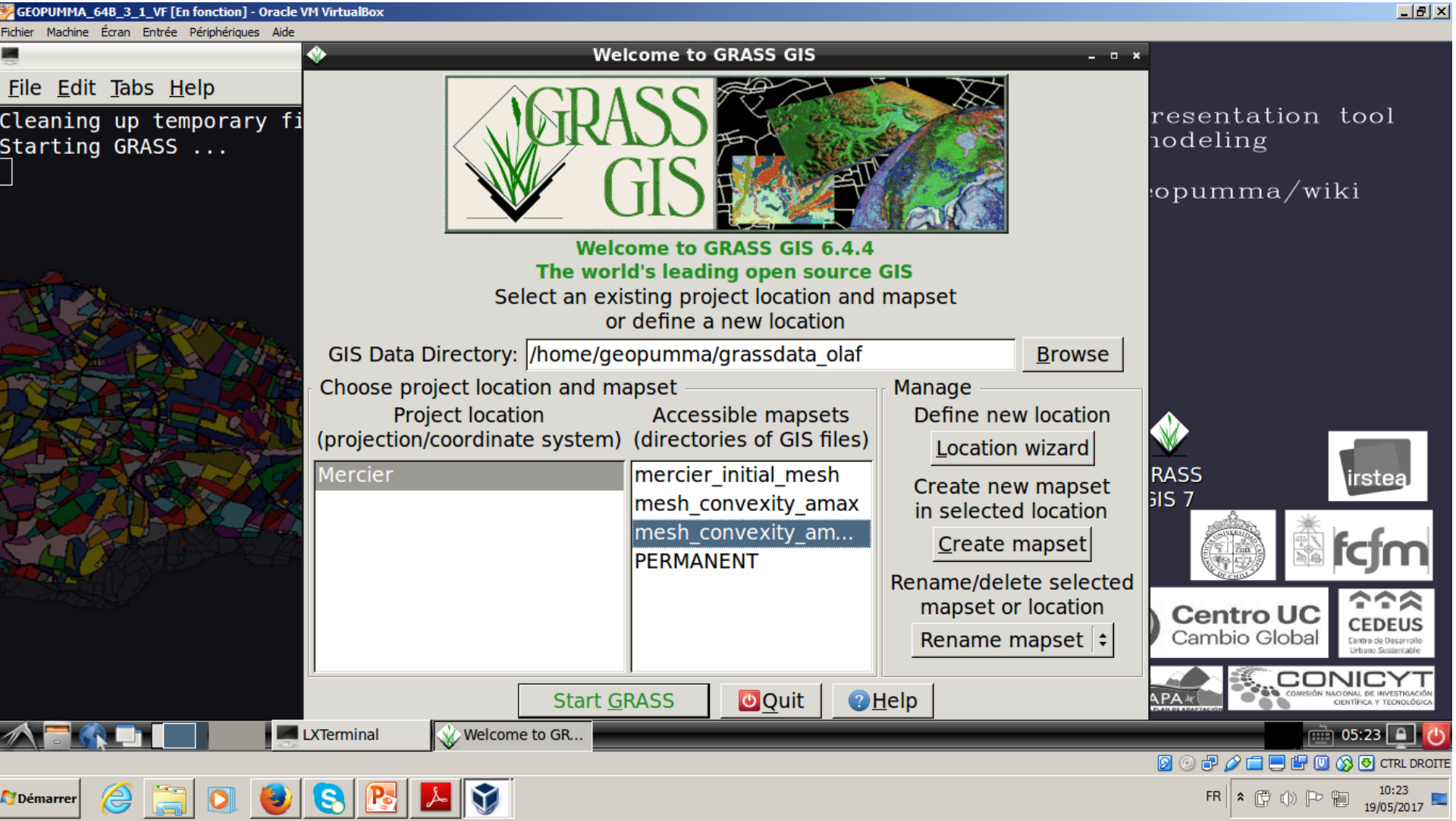
Step: Flow routing of the initial mesh

Visualize the vectorial layers corresponding to the surface drainage network in GRASS

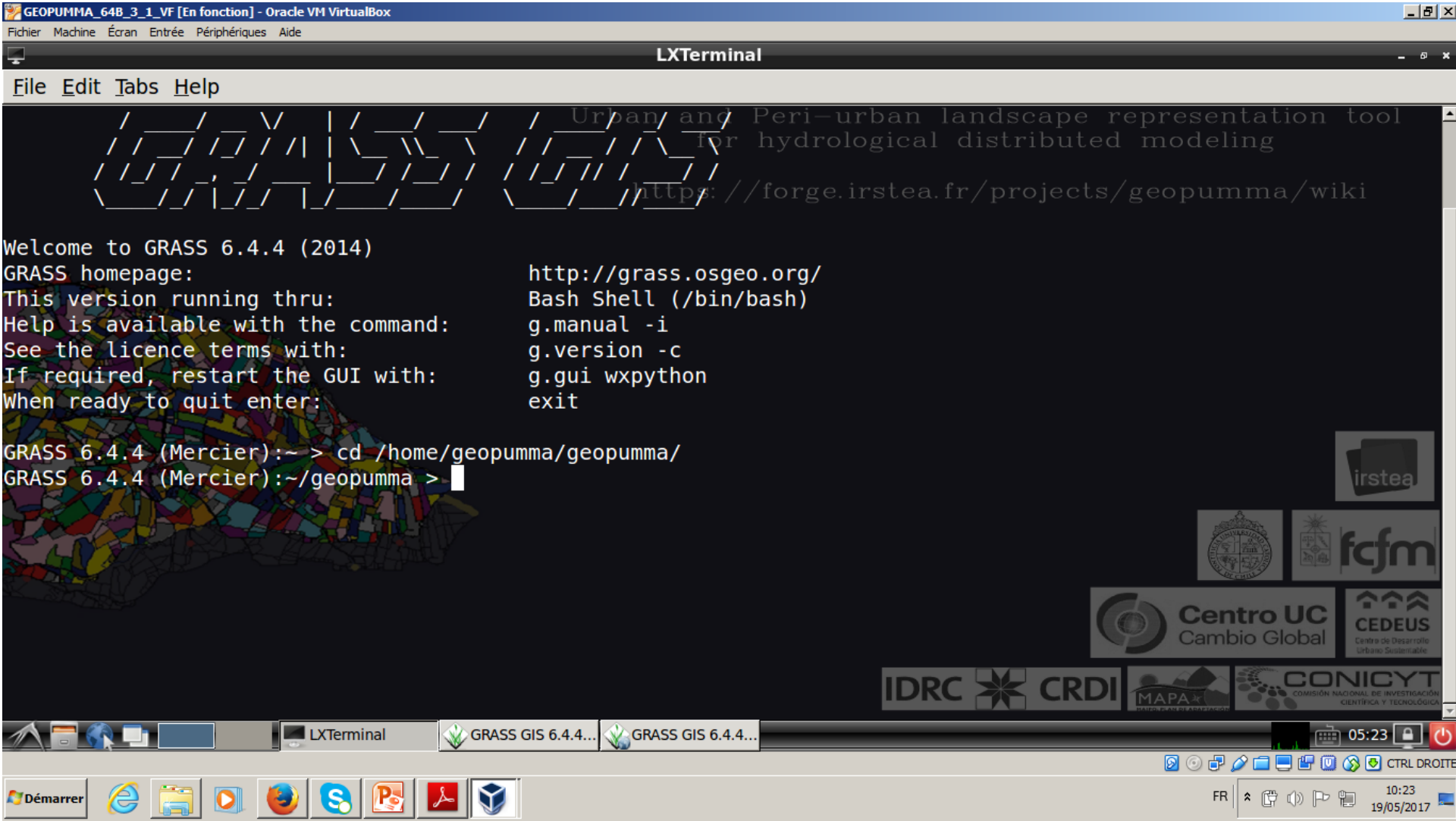
These elements belong to the drainage network shown in Figure x.37.c



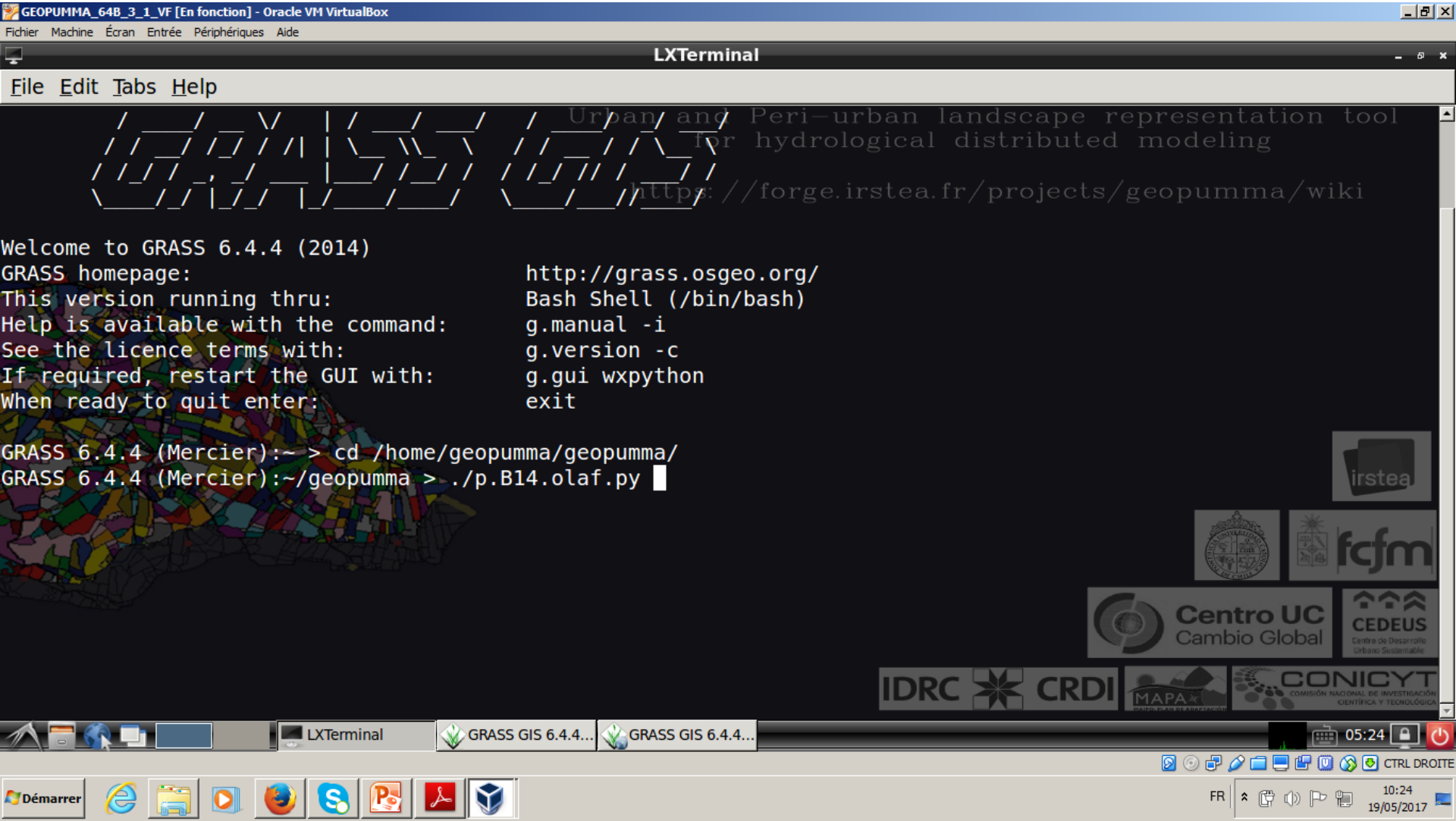
Step: Flow routing of the initial mesh
Select LOCATION: Mercier and MAPSET: mercier_convexity_amax_ff



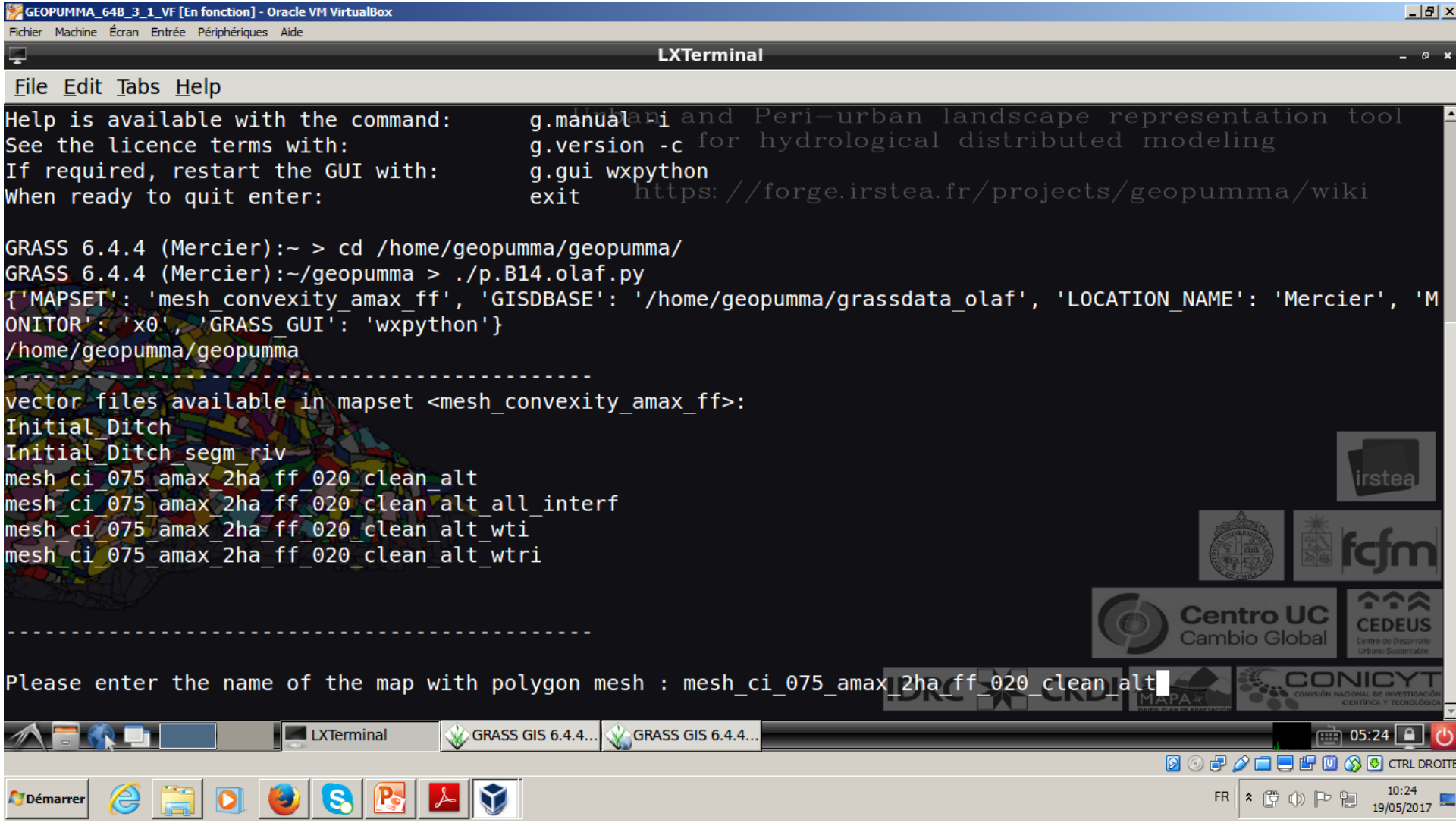
Step: Flow routing of the initial mesh
Change directory



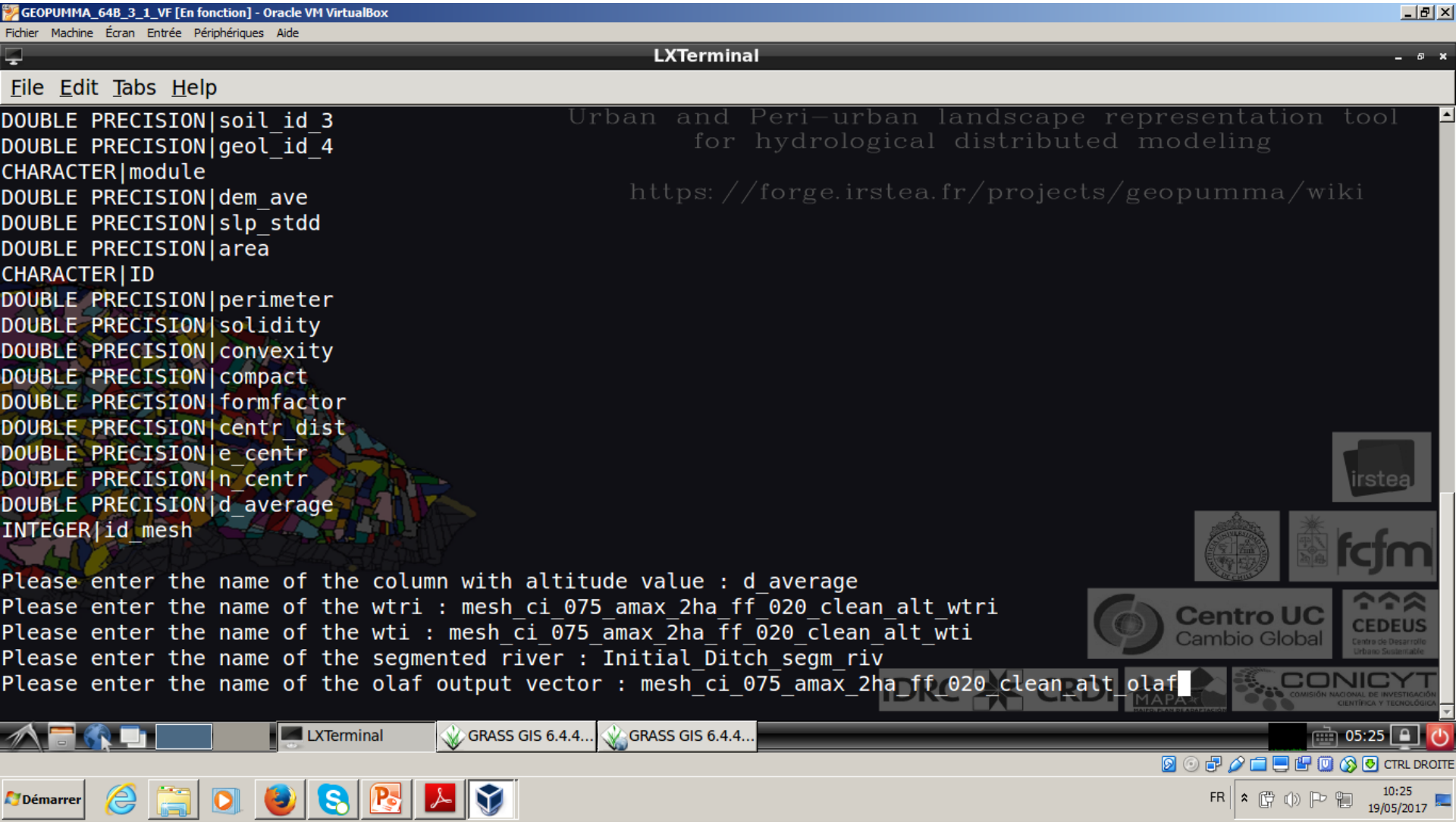
Step: Flow routing of the initial mesh
Execute the olaf command



Step: Flow routing of the initial mesh
Enter the following commands



Step: Flow routing of the initial mesh
Enter the following commands



Step: Flow routing of the initial mesh

Visualize the vectorial layers corresponding to the surface drainage network in GRASS

These elements belong to the drainage network shown in Figure x.37.d