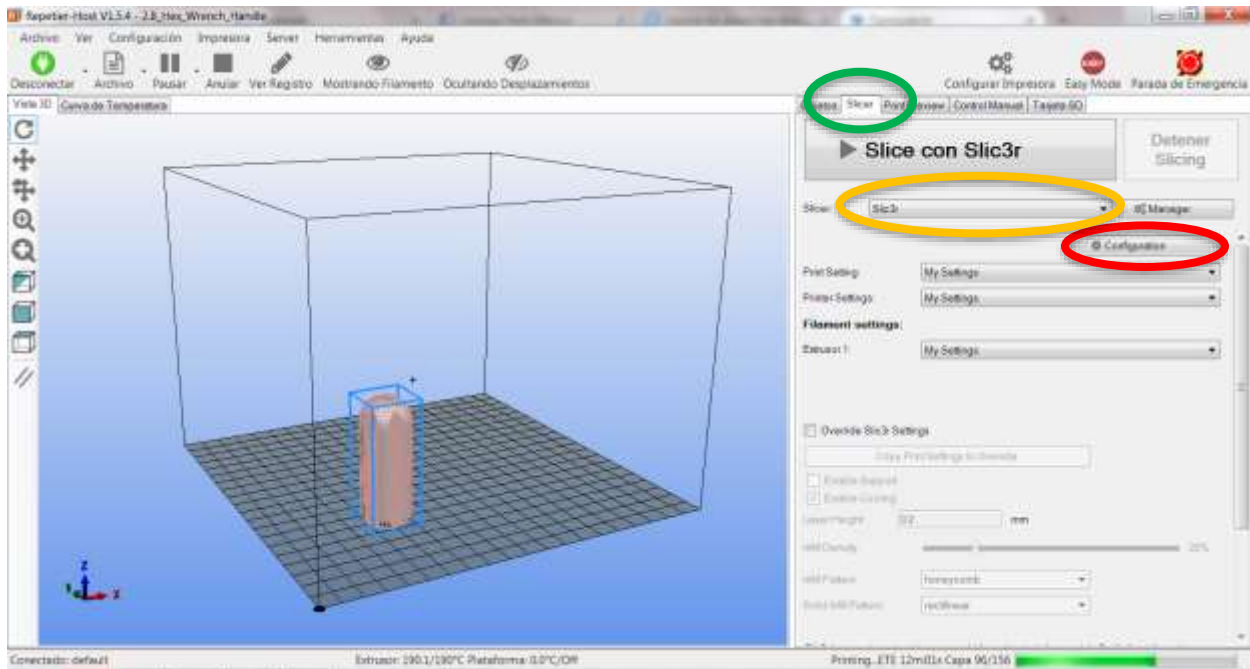
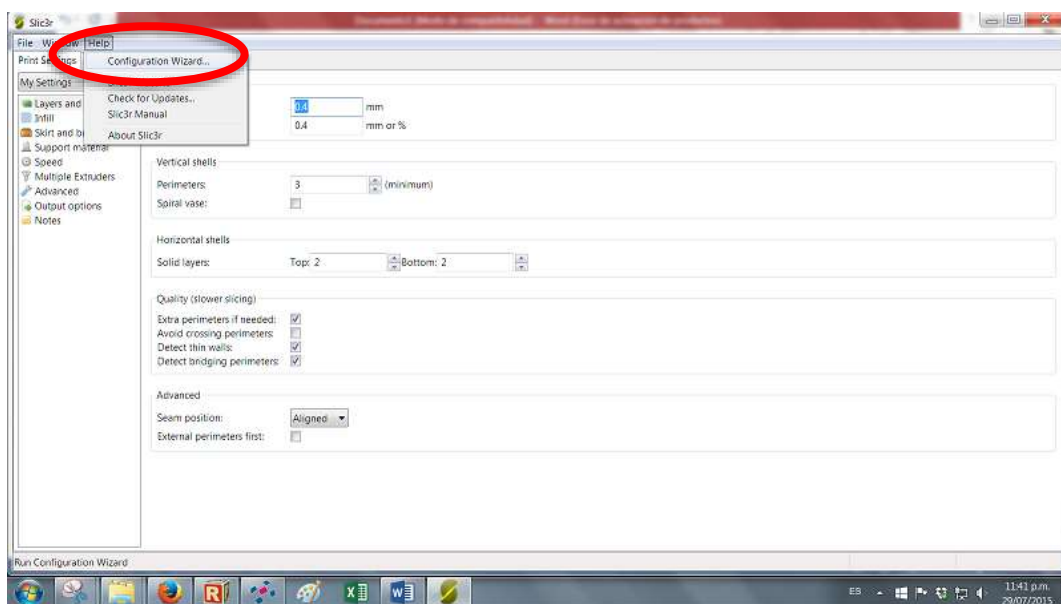


Configurando Slic3r con su "Wizard"

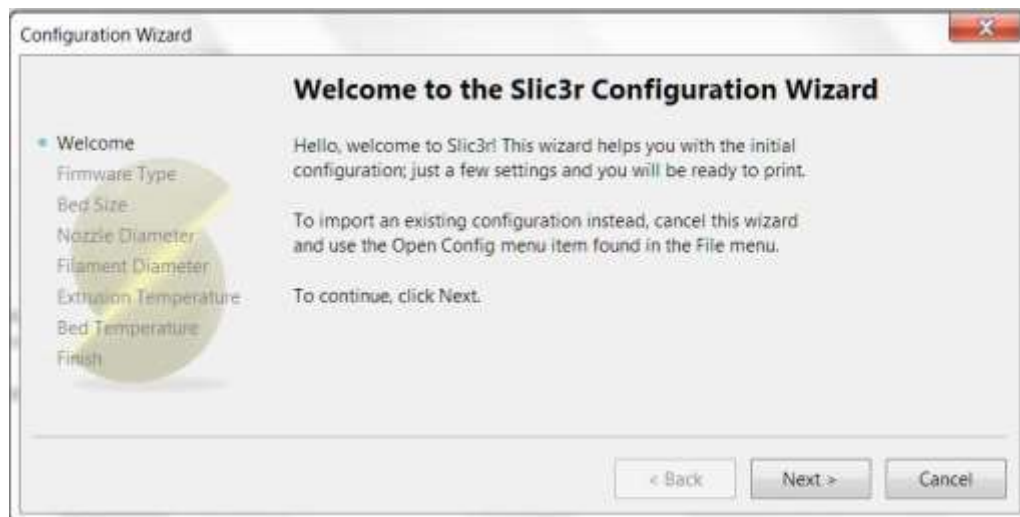
Primero vamos a ir a "Repetier Host" y desde su interfaz nos vamos a la pestaña de Slicer (verde), de ahí vamos a asegurarnos que estamos usando "Slic3r" como generador de código G (En naranja) y presionaremos el botón de "Configuración" (Rojo)



En la ventana de "Slic3r" vamos a ir a la parte de "Help" y de ahí a "configuration Wizard"



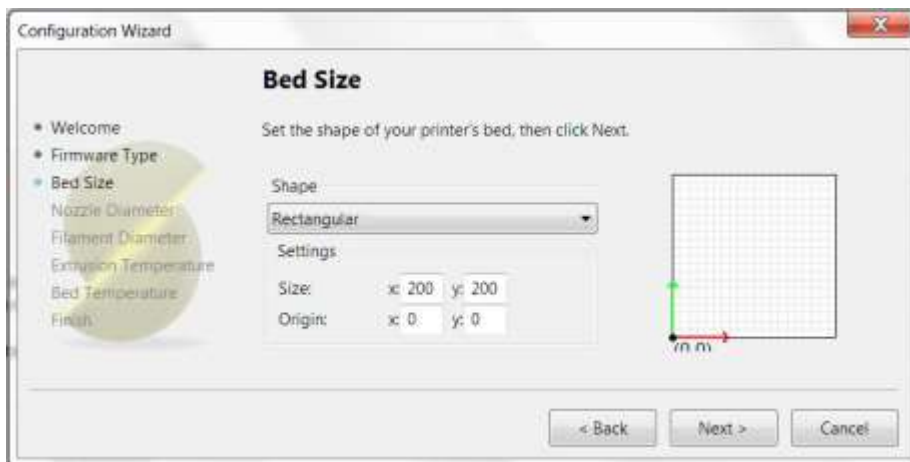
En el Wizard la primera ventana nos dara una bienvenida:



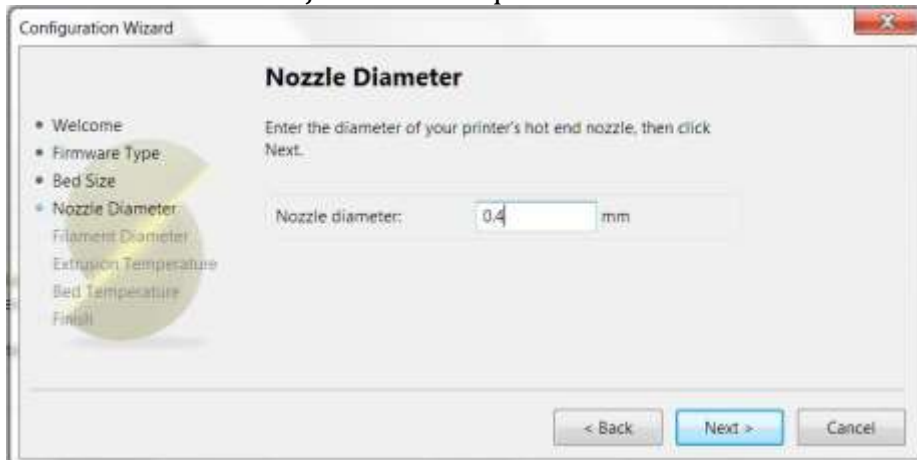
Firmware Type escogemos "RepRap(Marlin/Sprinter/Repetier)"



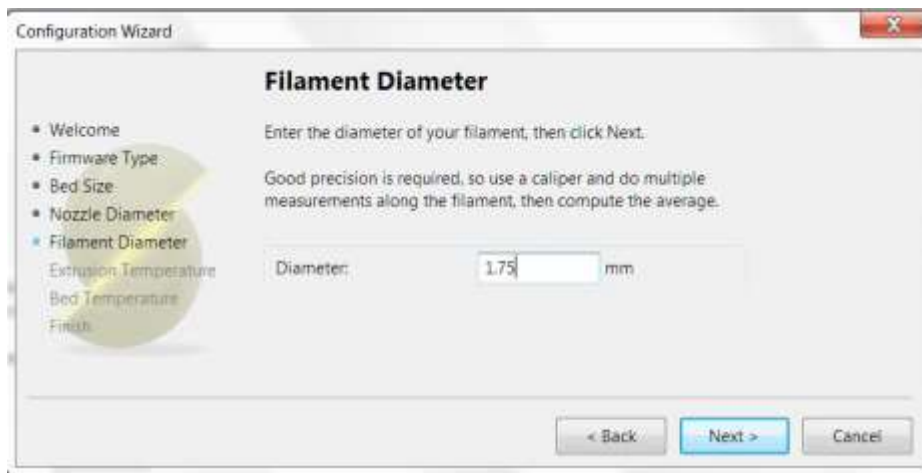
En Bed Size dejamos una forma Rectangular con tamaños 200x200mm y 0,0 en origen.



Nozzle Diameter lo dejamos en 0.4 que es el diámetro de salida de nuestra “boquilla”

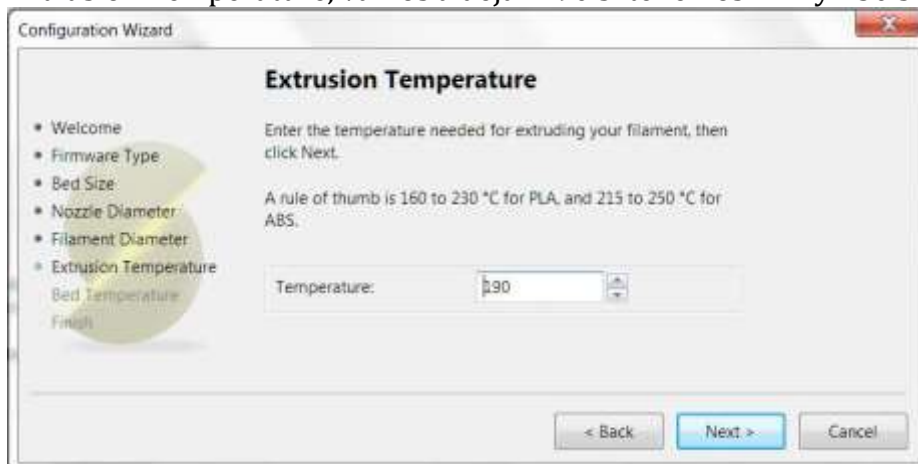


Filament Diameter vamos a poner el diámetro de nuestro filamento que es 1.75mm



The screenshot shows the 'Configuration Wizard' window with the 'Filament Diameter' step selected. The left sidebar lists the steps: Welcome, Firmware Type, Bed Size, Nozzle Diameter, Filament Diameter (highlighted), Extrusion Temperature, Bed Temperature, and Finish. The main area contains instructions: 'Enter the diameter of your filament, then click Next.' and 'Good precision is required, so use a caliper and do multiple measurements along the filament, then compute the average.' Below this is a text input field labeled 'Diameter:' with the value '1.75' and a unit dropdown set to 'mm'. At the bottom are buttons for '< Back', 'Next >', and 'Cancel'.

Extrusion Temperature, vamos a dejar 190 si tenemos PLA y 230 si tenemos ABS



The screenshot shows the 'Configuration Wizard' window with the 'Extrusion Temperature' step selected. The left sidebar lists the steps: Welcome, Firmware Type, Bed Size, Nozzle Diameter, Filament Diameter, Extrusion Temperature (highlighted), Bed Temperature, and Finish. The main area contains instructions: 'Enter the temperature needed for extruding your filament, then click Next.' and 'A rule of thumb is 160 to 230 °C for PLA, and 215 to 250 °C for ABS.' Below this is a temperature input field labeled 'Temperature:' with the value '190' and a unit dropdown set to '°C'. At the bottom are buttons for '< Back', 'Next >', and 'Cancel'.

La cama caliente la dejamos en 0 si tenemos PLA y 70 si tenemos ABS



The screenshot shows the 'Configuration Wizard' window with the 'Bed Temperature' step selected. The left sidebar lists the steps: Welcome, Firmware Type, Bed Size, Nozzle Diameter, Filament Diameter, Extrusion Temperature, Bed Temperature (highlighted), and Finish. The main area contains instructions: 'Enter the bed temperature needed for getting your filament to stick to your heated bed, then click Next.' and 'A rule of thumb is 60 °C for PLA and 110 °C for ABS. Leave zero if you have no heated bed.' Below this is a temperature input field labeled 'Bed temperature:' with the value '0' and a unit dropdown set to '°C'. At the bottom are buttons for '< Back', 'Next >', and 'Cancel'.

Después de terminar con el “wizard”, recuerda ir a “Filament Settings” y mantener activado el ventilador siempre como la siguiente configuración.

