**Term Description:**  
Layer height determines the thickness of each printed layer

Infill density refers to how solid the inside of the print is.

Print speed determines how fast the print head moves while extruding material  
  
**Prerequisites.**Python Installation:Ensure that Python is installed on your system. You can download and install Python from python.org.  
Tkinter Library:Tkinter is included with Python's standard library, so you don't need to install it separately. However, if you're using a minimal installation of Python, you might need to install it. You can install Tkinter using the package manager for your operating system (e.g., apt-get, yum, brew).  
PrusaSlicer:Download and install PrusaSlicer from the Prusa3D website.Make sure you have the correct path to the prusa-slicer.exe in your script. Adjust the path if necessary.  
External Libraries:If you're using a minimal Python setup, you might need to install some standard libraries. However, for this specific script, you only need the libraries that come with the standard Python distribution:tkintersubprocessos

How to use and complete rundown-

**Creating the Application Window:**   
The application window is created using a library called Tkinter, which is used to make simple GUI applications in Python.

**Selecting a 3D Model File:**

There’s a button labeled "Select 3D Model". When you click this button, it opens a dialog where you can choose a file with the .stl extension (a common file format for 3D models).

The selected file’s name is displayed in the application.

**Setting Printing Parameters:**

The application has several fields where you can enter important settings for your 3D print:

Layer Height (mm): How thick each layer of the print should be.

Infill Density (%): How solid or hollow the inside of the print should be (0% is hollow, 100% is completely solid).

Print Speed (mm/s): How fast the printer should move while printing.

Print Quality Mode: A dropdown menu where you can select a predefined quality setting (Draft, Normal, High Quality).

**Rotating the Model:**

There are buttons to rotate the model along the X, Y, and Z axes:

CW means clockwise rotation.

CCW means counterclockwise rotation.

There is also a button to reset the rotations back to the original position.

**Slicing the Model:**

Slicing is the process of converting the 3D model into instructions that a 3D printer can follow.

The "Slice Model" button takes all the settings you’ve entered, processes the model, and saves the instructions as a .gcode file.

You’ll see a message indicating whether the slicing was successful and where the file is saved.

**Visual Feedback:**

Throughout the process, messages are displayed to guide you, such as confirming file selection, rotation actions, or errors in input values.

**How the Code Works**

Libraries Used:

Tkinter for the GUI.

filedialog for opening file dialogs.

subprocess for running external commands (like slicing the model).

os for basic file operations.

Main Functions:

select\_file(): Opens a dialog to select the 3D model file.

rotate\_model(axis, direction): Rotates the model along a specified axis.

reset\_rotation(): Resets the rotation angles to zero.

slice\_model(): Processes the model with the given parameters and generates the .gcode file.

Creating the GUI:

Various Tkinter widgets (buttons, labels, entry fields) are used to create the interface.

Frames are used to organize these widgets into sections.

**Practical Use/How to use**

* Start the Application:
* Run the script to open the application window.
* Select Your Model:
* Click "Select 3D Model" and choose your .stl file.
* Set Your Parameters:
* Enter the layer height, infill density, and print speed.
* Choose a print quality mode.
* Adjust Model Orientation:
* Use the rotation buttons if needed.
* Slice the Model:
* Click "Slice Model" to generate the .gcode file.
* Check the Output:

Follow the on-screen messages to see if the slicing was successful and where the file is saved.