Running the Macros

1. Change the Path if necessary:

solidworks\_path = r"C:\Program Files\SOLIDWORKS Corp\SOLIDWORKS\SLDWORKS.exe"

1. final\_automation\_file.py
2. In file: write\_macro\_args.py
3. Change the Version of Solidworks (2023/2022/2024). If necessary, change the path also.

Set swModel = swApp.NewDocument("C:\\ProgramData\\SOLIDWORKS\\SOLIDWORKS 2023\\templates\\Part.PRTDOT", 0, 0, 0)

1. You can simply create the folder in location. If you don’t have D drive, create folder anywhere and change the path

Dim savePath As String: savePath = "D:\\ANK\\My macros\\Blocks.SLDPRT"

1. You can also change the path as above.

 macro\_path = r"D:\AnK\My macros\Blocks\plate1.swb"

1. In file: writehole\_macro\_args.py
   1. Do the same changes as above