Make an online heroku account & remember its password as well , cause it would be used during deployment phase.

DOWNLOAD HEROKU CLI (WINDOWS 64 BIT)

DOWNLOAD GIT CLI FROM GIT WEBSITE

INSTALL BOTH OF THEM , BY JUST CLICKING NEXT

OPEN ANACONDA PROMPT, WRITE COMMAND - CONDA CREATE –N FLASK\_APP\_DEMO PYTHON==3.7

(INSTEAD OF WRITING FLASK\_APP\_DEMO , U CAN GIVE ANY NAME OF ENVIRONMENT, SAME WITH THE

PYTHON VERSION)

CLICK YES TO PROCEED

CONDA ACTIVATE FLASK\_APP\_DEMO

THEN U WILL SEE FLASK\_APP\_DEMO IN STARTING OF COMMAND PROMPT, IT MEANS YOU ARE IN THE

ENVIRONMENT.

NOW OPEN PYCHARM, CLICK NEW PROJECT, GIVE NAME OF YOUR CHOICE LIKE FLASK\_APP\_INEURON,

EXPAND THE PYTHON INTERPRATOR TAB IF NOT EXPANDED, THEN CLICK ON PREVIOUSLY CONFIGURED

INTERPRETOR, THEN CLICK ON 3 DOTS IN THE NEXT LINE , IN NEW WINDOW CLICK ON CONDA ENVIRONMENT, AUTOMATICALLY THE COMMAND PATH WILL COME IN THE INTERPRATOR TAB, WHICH WOULD BE LIKE C/USER/USERS/ENVS/FLASK\_APP\_DEMO/PYTHON.EXE

BUT IF IT’S NOT COME, THEN GO TO 3 DOTS AT THE VERY RIGHT IN FRONT OF INTERPRATOR, THERE U CAN SEE MANY ENVIRONMENTS PREVIOULY CREATED BY YOU IN THE ENVS FOLDER, CLICK YOUR ENV NAME , THEN AT DOWNSIDE CLICK ON PYTHON.EXE, THEN OK,

PATH WILL BE SHOWED IN INTERPRATOR FOLDER, CLICK OK, THEN CREATE , CLICK THIS WINDOW,

CLICK ON TERMINAL IN PYCHARM, & THE ENV U SELECTED WILL BE VISIBLE IN THE STARTING.

IF IT’S NOT , U CAN CHECK BY CLICK FILE, THEN SETTINGS, THEN NAME OF THE PROJECT ( IN OUR CASE IT IS FLASK\_APP\_DEMO) , U WILL HAVE AN OPTION PYTHON INTERPRATER, THE NAME IS SELECTED THERE, IF NOT THEN CLICK ON ITS SETTINGS, CLICK ADD, IN THE POP UP WINDOW SELECT CONDA ENV,

THEN PYTHON VERSION LIKE 3.7 OR 3.6 WHICHEVER U HAVE SPECIFIED EARLIAR, ALL THIS IS DONE EITHER IN NEW ENVIRONMENT OPTION . BETTER IS U SELECT EXISTING INTERPTATOR, THEN 3 DOTS, GO TO ENVS, SELECT ENVIRONMENT NAME & PYTHON.EXE, THEN OK. OK. THEN CLOSE THE TERMINAL REOPEN IT, U WILL SEE THE ENVIRONMENT NAME IN STARTING OF COMMAND PROMPT.

FLASK\_APP\_DEMO IS THE NAME OF MY ENVIRONMENT, & FLASK\_APP\_INEURON IS THE NAME OF MY PROJECT, then in terminal write pip install flask to install flask.

Then remove previous code, & write your own code (your app code) . After that we run that code & from terminal click the link given, which automatically prompts us to browser where we saw our code execution. So we can say that code is running fine.

pip install gunicorn

Gunicorn is a liabrary that allows us to run our application in linux environment. As we are working in windows , so we have to install gunicorn because heroku works with linux based environments.

After that we write a new command in terminal as: pip freeze>requirements.txt .( don’t change the name of file ) After it’s execution you can see that a new file appears under main.py file with the name – requirements.txt containing about 12 short lines like

certify = 2021.02

Click = ……..

Color= …….

Etc etc

Now we make new file – go to flask project named folder - new – file give name to the file Procfile (name is case sensitive), then just press ok in next popup window(this file is without any extension . then go to the file (under main.py tab) , & write code - web: gunicorn main: app

Here main is the name of our file & app is the name of our flask application. Then open your git account , go to add repository, give it a name like heroku\_demo, u can add something in the description, like this is my first deployment (heroku), after creating the repository, u will see some code lines under code tab, from there you have to copy first command called – git init , & paste it in pycharm terminal, then copy second command – git add . , & execute in pycharm . then 3rd command as git commit –m “first commit”. If you are facing an error like author identity unknown then you have to stop & open the path folder in which main, Procfile etc all are given, then we open windows powershell there which have the same path in which your root folder is , & there u execute command 1St , 2nd & 3rd , in 3rd you will face previous issue, so u have to execute 2 commands given in your error like git config – global , in these commands you have to give your email id(in github) in first command & your username(your git hub username). After execution git will recognize your id & allow your system to connect with git. Now go to Git account there you will see the commands which I will use now in pycharm, you can just copy paste them or enter them manually. First run command - echo “# flask\_demo” >> README.md (here flask\_demo is name of my file, you must change it with your)run all the 3 command form starting, then execute – git branch –M main( here we are creating a branch named main). Then git remote add origin …….( you have to specify path) & then git push –u origin main. All these command also be copied form git & paste in windows powershell or u can also execute these commands in pycharm terminal. Your all code now uploaded on git , u can check it.

Then we open our heroku account & click on new , give name to your app, then click create app, go to deploy tab, checking below there under deployment method , press GitHub ,then downwards u find commands , copy heroku login –i & paste in pycharm & run , prompt appear asking your heroku mail id & then password, once u login successfully you will see message like logged in as [yogigautam83@gmail.com](mailto:yogigautam83@gmail.com) (its my id so I see this) , after then again prompt apppers , now go to heroku & copy the 3rd command from first code section(Create a new git repository) which would be like this – heroku git:remote a flask heroku app 035315, here the end no is your unique heroku no associated with your app. Now again go to heroku , there in 2nd code section named as deploy your application & copy 3rd line code & paste in pycharm terminal , it would be like – git push heroku master , replace master with main cause in our case master is not name of file but main is the name of file which contains the code.

Then you will see that deploy button click that & u see your code running . or you can go to app & click open app & you can see your application running in browser’s tab. Remember not to change the name of Procfile as procfile as it will result in error during app execution & throw error. There will be no error shown in deployment as it would show you deploy succeeded. You can see these errors in logs of the app, In that case just go to pycharm & change the name of file procfile to Procfile or go to root folder there change name from procfile to Procfile. After that u will see that in pycharm also the file name is changed. But it’s just to remember your mistake, the error will not resolve with that because even after changing the file name , the file name on git is not changed which is the root cause for that you have to go to git, there click on procfile & change name of file as Procfile, even after that error will not resolve. So you have to make a new app on heroku & then deploy using previous commands & your app will run .

(We can also make a new branch let say main1 (other than main) in GitHub & use the same code in that & then can use that for deployment , in that case we have to again use command git push heroku main1 which we have earliar used & also other commands to set up connection between git & heroku via pycharm & successfully upload our python code to git. But now the procfile name is changed associated with main1 file, so no error will come during deployment) Also u can do one more thing that erase the git repo & again make a new repo . this new repo contain file named Procfile , so there would be no problem in running the app after deployment . So there are many ways but the short one is I think changing name of file in git & make a new app.

(An error occurred while opening the first deployed app i.e. Application error, while searching for the errors in view logs , I found : at=error code=H14 desc="No web processes running" ………….

So this was due a small mistake where I wrongly named a file procfile, the right name is Procfile. Then I go to git, there in heroku\_demo repository , there are the all files which I submitted from pycharm. There I go to procfile , open it, then changed its name. Before that I tried a lot in pycharm to do that, first I change the name of procfile to Procfile, then I use: git push heroku main in pycharm terminal(in my venv)but it shows everything is fine, but when I saw in git, the file name was not changed from procfile to Procfile. So I changed it directly in git. Then I go to heroku , there again press new & make a new name of app called secondflaskdeploymentattempt. After that follow steps as per heroku, like I go to deploy, there I click github link, then authorize heroku to source code from git, then press deploy, then like in first attempt , I deployed from pycharm terminal , whatever execution it shows there, same was shown in heroku, & then when I view , I got my app running.)