# Prompt&input

usr\_requirment: "You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig."

max\_loop: 20

temperature: 0.01

batchsize: 10

searchdocs: 2

run\_times: 10

alpha\_PATH: "./workspace"

OPENAI\_API\_KEY: "sk-13a6fbf5f4894cd0877f12eb3eea98c2"

#OPENAI\_PROXY: "XXX"

OPENAI\_BASE\_URL: "https://api.deepseek.com/v1"

model: "deepseek-chat"

# Embedding

(ximualpha) root@ubuntu:/data/sda/lichenshuo/XiMuAlpha4CFD# ./run\_pipeline.sh

Please select an input file from the list below:

1) BuoyantCavity\_0\_Simple.yaml 5) Combustion.yaml 9) PitzDaily.yaml

2) BuoyantCavity\_pre.yaml 6) CylinderFlow.yaml 10) Planar\_Poiseuille.yaml

3) Cavity.yaml 7) CylinderFlow\_0.yaml 11) SquareBendLiq.yaml

4) Cavity\_RANS.yaml 8) HIT.yaml

#? 7

You selected CylinderFlow\_0.yaml

Running config\_path.py to load system paths...

config\_file\_path inputs/CylinderFlow\_0.yaml

Configuration loaded successfully:

usr\_requirment: You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.

./workspace/config/config2.yaml has been updated successfully.

Running Tutorial\_postprocess.py for data preprocessing...

config\_file\_path inputs/CylinderFlow\_0.yaml

Configuration loaded successfully:

usr\_requirment: You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.

./workspace/config/config2.yaml has been updated successfully.

Traceback (most recent call last):

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/Tutorial\_postprocess.py", line 228, in <module>

raise EnvironmentError("The environment variable WM\_PROJECT\_DIR is not set.")

OSError: The environment variable WM\_PROJECT\_DIR is not set.

Running Langchain\_database\_add\_tutorial\_summary.py...

config\_file\_path inputs/CylinderFlow\_0.yaml

Configuration loaded successfully:

usr\_requirment: You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.

./workspace/config/config2.yaml has been updated successfully.

i: 0

i: 10

i: 20

i: 30

i: 40

i: 50

i: 60

i: 70

i: 80

i: 90

i: 100

i: 110

i: 120

i: 130

i: 140

i: 150

i: 160

i: 170

i: 180

i: 190

i: 200

i: 210

i: 220

i: 230

i: 240

i: 250

i: 260

i: 270

i: 280

i: 290

i: 300

i: 310

i: 320

i: 330

i: 340

i: 350

i: 360

i: 370

i: 380

i: 390

i: 400

i: 410

i: 420

i: 430

i: 440

i: 450

i: 460

i: 470

i: 480

i: 490

i: 500

i: 510

i: 520

i: 530

i: 540

i: 550

i: 560

i: 570

Running Langchain\_database\_add\_tutorial.py...

config\_file\_path inputs/CylinderFlow\_0.yaml

Configuration loaded successfully:

usr\_requirment: You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.

./workspace/config/config2.yaml has been updated successfully.

i: 0

i: 10

i: 20

i: 30

i: 40

i: 50

i: 60

i: 70

i: 80

i: 90

i: 100

i: 110

i: 120

i: 130

i: 140

i: 150

i: 160

i: 170

i: 180

i: 190

i: 200

i: 210

i: 220

i: 230

i: 240

i: 250

i: 260

i: 270

i: 280

i: 290

i: 300

i: 310

i: 320

i: 330

i: 340

i: 350

i: 360

i: 370

i: 380

i: 390

i: 400

i: 410

i: 420

i: 430

i: 440

i: 450

i: 460

i: 470

i: 480

i: 490

i: 500

i: 510

i: 520

i: 530

i: 540

i: 550

i: 560

i: 570

i: 580

i: 590

i: 600

i: 610

i: 620

i: 630

i: 640

i: 650

i: 660

i: 670

i: 680

i: 690

i: 700

i: 710

i: 720

i: 730

i: 740

i: 750

i: 760

i: 770

i: 780

i: 790

i: 800

i: 810

i: 820

i: 830

i: 840

i: 850

i: 860

i: 870

i: 880

i: 890

i: 900

i: 910

i: 920

i: 930

i: 940

i: 950

i: 960

i: 970

i: 980

i: 990

i: 1000

i: 1010

i: 1020

i: 1030

i: 1040

i: 1050

i: 1060

i: 1070

i: 1080

i: 1090

i: 1100

i: 1110

i: 1120

i: 1130

i: 1140

i: 1150

i: 1160

i: 1170

i: 1180

i: 1190

i: 1200

i: 1210

i: 1220

i: 1230

i: 1240

i: 1250

i: 1260

i: 1270

i: 1280

i: 1290

i: 1300

i: 1310

i: 1320

i: 1330

i: 1340

i: 1350

i: 1360

i: 1370

i: 1380

i: 1390

i: 1400

i: 1410

i: 1420

i: 1430

i: 1440

i: 1450

i: 1460

i: 1470

i: 1480

i: 1490

i: 1500

i: 1510

i: 1520

i: 1530

i: 1540

i: 1550

i: 1560

i: 1570

i: 1580

i: 1590

i: 1600

i: 1610

i: 1620

i: 1630

i: 1640

i: 1650

i: 1660

i: 1670

i: 1680

i: 1690

i: 1700

i: 1710

i: 1720

i: 1730

i: 1740

i: 1750

i: 1760

i: 1770

i: 1780

i: 1790

i: 1800

i: 1810

i: 1820

i: 1830

i: 1840

i: 1850

i: 1860

i: 1870

i: 1880

i: 1890

i: 1900

i: 1910

i: 1920

i: 1930

i: 1940

i: 1950

i: 1960

i: 1970

i: 1980

i: 1990

i: 2000

i: 2010

i: 2020

i: 2030

i: 2040

i: 2050

i: 2060

i: 2070

i: 2080

i: 2090

i: 2100

i: 2110

i: 2120

i: 2130

i: 2140

i: 2150

i: 2160

i: 2170

i: 2180

i: 2190

i: 2200

i: 2210

i: 2220

i: 2230

i: 2240

i: 2250

i: 2260

i: 2270

i: 2280

i: 2290

i: 2300

i: 2310

i: 2320

i: 2330

i: 2340

i: 2350

i: 2360

i: 2370

i: 2380

i: 2390

i: 2400

i: 2410

i: 2420

i: 2430

i: 2440

i: 2450

i: 2460

i: 2470

i: 2480

i: 2490

i: 2500

i: 2510

i: 2520

i: 2530

i: 2540

i: 2550

i: 2560

i: 2570

i: 2580

i: 2590

i: 2600

i: 2610

i: 2620

i: 2630

i: 2640

i: 2650

i: 2660

i: 2670

i: 2680

i: 2690

i: 2700

i: 2710

i: 2720

i: 2730

i: 2740

i: 2750

i: 2760

i: 2770

i: 2780

i: 2790

i: 2800

i: 2810

i: 2820

i: 2830

i: 2840

i: 2850

i: 2860

i: 2870

i: 2880

i: 2890

i: 2900

i: 2910

i: 2920

i: 2930

i: 2940

i: 2950

i: 2960

i: 2970

i: 2980

i: 2990

i: 3000

i: 3010

i: 3020

i: 3030

i: 3040

i: 3050

i: 3060

i: 3070

i: 3080

i: 3090

i: 3100

i: 3110

i: 3120

i: 3130

i: 3140

i: 3150

i: 3160

i: 3170

i: 3180

i: 3190

i: 3200

i: 3210

i: 3220

i: 3230

i: 3240

i: 3250

i: 3260

i: 3270

i: 3280

i: 3290

i: 3300

i: 3310

i: 3320

i: 3330

i: 3340

i: 3350

i: 3360

i: 3370

i: 3380

i: 3390

i: 3400

i: 3410

i: 3420

i: 3430

i: 3440

i: 3450

i: 3460

i: 3470

i: 3480

i: 3490

i: 3500

i: 3510

i: 3520

i: 3530

i: 3540

i: 3550

i: 3560

i: 3570

i: 3580

i: 3590

i: 3600

i: 3610

i: 3620

i: 3630

i: 3640

i: 3650

i: 3660

i: 3670

i: 3680

i: 3690

i: 3700

i: 3710

i: 3720

i: 3730

i: 3740

i: 3750

i: 3760

i: 3770

i: 3780

i: 3790

i: 3800

i: 3810

i: 3820

i: 3830

i: 3840

i: 3850

i: 3860

i: 3870

i: 3880

i: 3890

i: 3900

i: 3910

i: 3920

i: 3930

i: 3940

i: 3950

i: 3960

i: 3970

i: 3980

i: 3990

i: 4000

i: 4010

i: 4020

i: 4030

i: 4040

i: 4050

i: 4060

i: 4070

i: 4080

i: 4090

i: 4100

i: 4110

i: 4120

i: 4130

i: 4140

i: 4150

i: 4160

i: 4170

i: 4180

i: 4190

i: 4200

i: 4210

i: 4220

i: 4230

i: 4240

i: 4250

i: 4260

i: 4270

i: 4280

i: 4290

i: 4300

i: 4310

i: 4320

i: 4330

i: 4340

i: 4350

i: 4360

i: 4370

i: 4380

i: 4390

i: 4400

i: 4410

i: 4420

i: 4430

i: 4440

i: 4450

i: 4460

i: 4470

i: 4480

i: 4490

i: 4500

i: 4510

i: 4520

i: 4530

i: 4540

i: 4550

i: 4560

i: 4570

i: 4580

i: 4590

i: 4600

i: 4610

i: 4620

i: 4630

i: 4640

i: 4650

i: 4660

i: 4670

i: 4680

i: 4690

i: 4700

i: 4710

i: 4720

i: 4730

i: 4740

i: 4750

i: 4760

i: 4770

i: 4780

i: 4790

i: 4800

i: 4810

i: 4820

i: 4830

i: 4840

i: 4850

i: 4860

i: 4870

i: 4880

i: 4890

i: 4900

i: 4910

i: 4920

i: 4930

i: 4940

i: 4950

i: 4960

i: 4970

i: 4980

i: 4990

i: 5000

i: 5010

i: 5020

i: 5030

i: 5040

i: 5050

i: 5060

i: 5070

i: 5080

i: 5090

i: 5100

i: 5110

i: 5120

i: 5130

i: 5140

i: 5150

i: 5160

i: 5170

i: 5180

i: 5190

i: 5200

i: 5210

i: 5220

i: 5230

i: 5240

i: 5250

i: 5260

i: 5270

i: 5280

i: 5290

i: 5300

i: 5310

i: 5320

i: 5330

i: 5340

i: 5350

i: 5360

i: 5370

i: 5380

i: 5390

i: 5400

i: 5410

i: 5420

i: 5430

i: 5440

i: 5450

i: 5460

i: 5470

i: 5480

i: 5490

i: 5500

i: 5510

i: 5520

i: 5530

i: 5540

i: 5550

i: 5560

i: 5570

i: 5580

i: 5590

i: 5600

i: 5610

i: 5620

i: 5630

i: 5640

i: 5650

i: 5660

i: 5670

i: 5680

i: 5690

i: 5700

i: 5710

i: 5720

i: 5730

i: 5740

i: 5750

i: 5760

i: 5770

i: 5780

i: 5790

i: 5800

i: 5810

i: 5820

i: 5830

i: 5840

i: 5850

i: 5860

i: 5870

i: 5880

i: 5890

i: 5900

i: 5910

i: 5920

i: 5930

i: 5940

i: 5950

i: 5960

i: 5970

i: 5980

i: 5990

i: 6000

i: 6010

i: 6020

i: 6030

i: 6040

i: 6050

i: 6060

i: 6070

i: 6080

i: 6090

i: 6100

i: 6110

i: 6120

i: 6130

i: 6140

i: 6150

i: 6160

i: 6170

i: 6180

i: 6190

i: 6200

i: 6210

i: 6220

i: 6230

i: 6240

i: 6250

i: 6260

i: 6270

i: 6280

i: 6290

i: 6300

i: 6310

i: 6320

i: 6330

i: 6340

i: 6350

i: 6360

i: 6370

i: 6380

i: 6390

i: 6400

i: 6410

i: 6420

i: 6430

i: 6440

i: 6450

i: 6460

i: 6470

i: 6480

i: 6490

i: 6500

i: 6510

i: 6520

i: 6530

i: 6540

i: 6550

i: 6560

i: 6570

i: 6580

i: 6590

i: 6600

i: 6610

i: 6620

i: 6630

i: 6640

i: 6650

i: 6660

i: 6670

i: 6680

i: 6690

i: 6700

i: 6710

i: 6720

i: 6730

i: 6740

i: 6750

i: 6760

i: 6770

i: 6780

i: 6790

i: 6800

i: 6810

i: 6820

i: 6830

i: 6840

i: 6850

i: 6860

i: 6870

i: 6880

i: 6890

i: 6900

i: 6910

i: 6920

i: 6930

i: 6940

i: 6950

i: 6960

i: 6970

i: 6980

i: 6990

i: 7000

i: 7010

i: 7020

i: 7030

i: 7040

i: 7050

i: 7060

i: 7070

i: 7080

i: 7090

i: 7100

i: 7110

i: 7120

i: 7130

i: 7140

i: 7150

i: 7160

i: 7170

i: 7180

i: 7190

i: 7200

i: 7210

i: 7220

i: 7230

i: 7240

i: 7250

i: 7260

i: 7270

i: 7280

i: 7290

i: 7300

i: 7310

i: 7320

i: 7330

i: 7340

i: 7350

i: 7360

i: 7370

i: 7380

i: 7390

i: 7400

i: 7410

i: 7420

i: 7430

i: 7440

i: 7450

i: 7460

i: 7470

i: 7480

i: 7490

i: 7500

i: 7510

i: 7520

i: 7530

i: 7540

i: 7550

i: 7560

i: 7570

i: 7580

i: 7590

i: 7600

i: 7610

i: 7620

i: 7630

i: 7640

i: 7650

i: 7660

i: 7670

i: 7680

i: 7690

i: 7700

i: 7710

i: 7720

i: 7730

i: 7740

i: 7750

i: 7760

i: 7770

i: 7780

i: 7790

i: 7800

i: 7810

i: 7820

i: 7830

i: 7840

i: 7850

i: 7860

i: 7870

i: 7880

i: 7890

i: 7900

i: 7910

i: 7920

i: 7930

i: 7940

i: 7950

i: 7960

i: 7970

i: 7980

i: 7990

i: 8000

i: 8010

i: 8020

i: 8030

i: 8040

i: 8050

i: 8060

i: 8070

i: 8080

i: 8090

i: 8100

i: 8110

i: 8120

i: 8130

i: 8140

i: 8150

i: 8160

i: 8170

i: 8180

i: 8190

i: 8200

i: 8210

i: 8220

i: 8230

i: 8240

i: 8250

i: 8260

i: 8270

i: 8280

i: 8290

i: 8300

i: 8310

i: 8320

i: 8330

i: 8340

Running Langchain\_database\_add\_command.py...

config\_file\_path inputs/CylinderFlow\_0.yaml

Configuration loaded successfully:

usr\_requirment: You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.

./workspace/config/config2.yaml has been updated successfully.

i: 0

i: 10

i: 20

i: 30

i: 40

i: 50

i: 60

i: 70

i: 80

i: 90

i: 100

i: 110

i: 120

i: 130

i: 140

i: 150

i: 160

i: 170

i: 180

i: 190

i: 200

i: 210

i: 220

i: 230

i: 240

i: 250

i: 260

i: 270

i: 280

i: 290

i: 300

Running Langchain\_database\_add\_allrun.py...

config\_file\_path inputs/CylinderFlow\_0.yaml

Configuration loaded successfully:

usr\_requirment: You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.

./workspace/config/config2.yaml has been updated successfully.

i: 0

i: 10

i: 20

i: 30

i: 40

i: 50

i: 60

i: 70

i: 80

i: 90

i: 100

i: 110

i: 120

i: 130

i: 140

i: 150

i: 160

i: 170

i: 180

i: 190

i: 200

i: 210

i: 220

i: 230

i: 240

i: 250

i: 260

i: 270

i: 280

i: 290

i: 300

i: 310

i: 320

i: 330

i: 340

i: 350

i: 360

i: 370

i: 380

i: 390

i: 400

i: 410

i: 420

i: 430

i: 440

i: 450

i: 460

i: 470

i: 480

i: 490

i: 500

i: 510

i: 520

i: 530

i: 540

i: 550

i: 560

i: 570

# Run

## Start, usr\_requirment，runtimes: 1roles.Architect:\_act:26 - Zhuxu: to do ArchitectAction(ArchitectAction)

Running alphaOpenfoam\_v2.py to execute the main program...

config\_file\_path inputs/CylinderFlow\_0.yaml

Configuration loaded successfully:

usr\_requirment: You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.

./workspace/config/config2.yaml has been updated successfully.

runtimes: 1

/root/anaconda3/envs/ximualpha/lib/python3.10/site-packages/langchain\_core/\_api/deprecation.py:117: LangChainDeprecationWarning: The class `langchain\_community.chat\_models.openai.ChatOpenAI` was deprecated in langchain-community 0.0.10 and will be removed in 0.2.0. An updated version of the class exists in the langchain-openai package and should be used instead. To use it run `pip install -U langchain-openai` and import as `from langchain\_openai import ChatOpenAI`.

warn\_deprecated(

2024-08-28 23:02:58.813 | INFO | roles.Architect:\_act:26 - Zhuxu: to do ArchitectAction(ArchitectAction)

self.rc.history: You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.

user\_case: case name: Cylinder\_Flow\_0

case domain: incompressible

case category: cylinder

case solver: overPimpleDyMFoam

find\_case page\_content="case name: cylinderAndBackground\ncase domain: incompressible\ncase category: cylinder\ncase solver: overPimpleDyMFoam\ncase input name:['U', 'pointDisplacement', 'cellDisplacement', 'zoneID', 'epsilon', 'nut', 'k', 'p', 'blockMeshDict', 'setFieldsDict', 'controlDict', 'topoSetDict', 'fvSchemes', 'decomposeParDict', 'fvSolution', 'dynamicMeshDict', 'turbulenceProperties', 'transportProperties']\ncorresponding input folder:{'U': '0.orig', 'pointDisplacement': '0.orig', 'cellDisplacement': '0.orig', 'zoneID': '0.orig', 'epsilon': '0.orig', 'nut': '0.orig', 'k': '0.orig', 'p': '0.orig', 'blockMeshDict': 'system', 'setFieldsDict': 'system', 'controlDict': 'system', 'topoSetDict': 'system', 'fvSchemes': 'system', 'decomposeParDict': 'system', 'fvSolution': 'system', 'dynamicMeshDict': 'constant', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}" metadata={'source': '/data/sda/lichenshuo/XiMuAlpha4CFD/database/openfoam\_tutorials\_summary.txt'}

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/find\_tutorial.txt

2024-08-28 23:05:46.963 | INFO | actions.ArchitectAction:run:111 - ```splits into 18 subtasks:

subtask1: to Write a OpenFoam U foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask2: to Write a OpenFoam pointDisplacement foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask3: to Write a OpenFoam cellDisplacement foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask4: to Write a OpenFoam zoneID foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask5: to Write a OpenFoam epsilon foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask6: to Write a OpenFoam nut foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask7: to Write a OpenFoam k foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask8: to Write a OpenFoam p foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask9: to Write a OpenFoam blockMeshDict foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask10: to Write a OpenFoam setFieldsDict foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask11: to Write a OpenFoam controlDict foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask12: to Write a OpenFoam topoSetDict foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask13: to Write a OpenFoam fvSchemes foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask14: to Write a OpenFoam decomposeParDict foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask15: to Write a OpenFoam fvSolution foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask16: to Write a OpenFoam dynamicMeshDict foamfile in constant folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask17: to Write a OpenFoam turbulenceProperties foamfile in constant folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..

subtask18: to Write a OpenFoam transportProperties foamfile in constant folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..```

2024-08-28 23:05:46.968 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Architect: 18

get\_memories\_InputWriter [user: to Write a OpenFoam U foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam pointDisplacement foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam cellDisplacement foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam zoneID foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam epsilon foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam nut foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam k foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam p foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam blockMeshDict foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam setFieldsDict foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam controlDict foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam topoSetDict foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam fvSchemes foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam decomposeParDict foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam fvSolution foamfile in system folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam dynamicMeshDict foamfile in constant folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam turbulenceProperties foamfile in constant folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig.., user: to Write a OpenFoam transportProperties foamfile in constant folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar foamfiles in tutorials that you find. 3. The generated content must not have the word empty. 4. For the 0.orig folder, name it 0 instead of 0.orig..```]

tutorial\_file: None

find\_similar\_foamfile: Based on the requirements provided, the closest matching OpenFOAM foamfile for the `U` field in the `0` folder for the case `Cylinder\_Flow\_0` would be similar to the `U` file from the `cylinderAndBackground` case, but adjusted to reflect the specific conditions and boundary types needed for a RANS simulation of flow around a cylinder using `overPimpleDyMFoam`. Here is a suggested `U` foamfile for the `0` folder of the `Cylinder\_Flow\_0` case:

```plaintext

/\*--------------------------------\*- C++ -\*----------------------------------\*\

| ========= | |

| \\ / F ield | OpenFOAM: The Open Source CFD Toolbox |

| \\ / O peration | Version: v2312 |

| \\ / A nd | Website: www.openfoam.com |

| \\/ M anipulation | |

\\*---------------------------------------------------------------------------\*/

FoamFile

{

version 2.0;

format ascii;

class volVectorField;

object U;

}

// \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* //

#include "include/initialConditions"

dimensions [0 1 -1 0 0 0 0];

internalField uniform (0.1 0 0); // Assuming a flow velocity of 0.1 m/s in the x-direction

boundaryField

{

#includeEtc "caseDicts/setConstraintTypes"

overset

{

type overset;

}

walls

{

type movingWallVelocity;

value uniform (0 0 0);

}

inlet

{

type fixedValue;

value uniform (0.1 0 0); // Consistent with internalField

}

outlet

{

type inletOutlet;

inletValue uniform (0 0 0);

value $internalField;

}

topAndBottom

{

type zeroGradient;

}

}

// \*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\* //

```

This file sets up the velocity field for a steady flow around a cylinder at a Reynolds number of 100, with a flow velocity of 0.1 m/s in the x-direction. The boundary conditions are adjusted to match the requirements of the simulation, including the use of `fixedValue` at the inlet and `inletOutlet` at the outlet. The `movingWallVelocity` type is used for the walls, and `zeroGradient` is used for the top and bottom boundaries.

find\_tutorial\_file: input p file of case cylinderAndBackground (domain: incompressible, category: cylinder, solver:overPimpleDyMFoam) in 0.orig folder:

/\*--------------------------------\*- C++ -\*----------------------------------\*\

| ========= | |

| \\ / F ield | OpenFOAM: The Open Source CFD Toolbox |

| \\ / O peration | Version: v2312 |

| \\ / A nd | Website: www.openfoam.com |

| \\/ M anipulation | |

\\*---------------------------------------------------------------------------\*/

FoamFile

{

version 2.0;

format ascii;

class volScalarField;

object p;

}

// \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* //

#include "include/initialConditions"

dimensions [0 2 -2 0 0 0 0];

internalField uniform $pressure;

boundaryField

{

#includeEtc "caseDicts/setConstraintTypes"

overset

{

type overset;

}

wall

{

type zeroGradient;

}

inlet

{

type zeroGradient;

}

outlet

{

type fixedValue; //calculated;

value $internalField;

}

".\*"

{

type zeroGradient;

}

}

// \*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\* //

File U foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar is going to be written

dict\_keys(['id', 'choices', 'created', 'model', 'object', 'system\_fingerprint', 'usage'])

folder\_name 0

file\_name U foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar

# 报错

2024-08-28 23:06:34.273 | WARNING | alpha.utils.common:wrapper:571 - There is a exception in role's execution, in order to resume, we delete the newest role communication message in the role's memory.

Traceback (most recent call last):

File "/root/anaconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/utils/common.py", line 562, in wrapper

return await func(self, \*args, \*\*kwargs)

File "/root/anaconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/roles/role.py", line 575, in run

rsp = await self.react()

File "/root/anaconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/roles/role.py", line 542, in react

rsp= await self.\_react()

File "/root/anaconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/roles/role.py", line 488, in \_react

rsp = await self.\_act()

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/roles/InputWriter.py", line 36, in \_act

code\_text = await todo.run(context)

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/actions/InputWriterAction.py", line 72, in run

self.save\_file(file\_path, code\_context=str(code\_text))

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/actions/InputWriterAction.py", line 182, in save\_file

with open(file\_path, 'w') as file:

OSError: [Errno 36] File name too long: '/data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/0/U foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar'

During handling of the above exception, another exception occurred:

Traceback (most recent call last):

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/alphaOpenfoam\_v2.py", line 74, in <module>

asyncio.run(main())

File "/root/anaconda3/envs/ximualpha/lib/python3.10/asyncio/runners.py", line 44, in run

return loop.run\_until\_complete(main)

File "/root/anaconda3/envs/ximualpha/lib/python3.10/asyncio/base\_events.py", line 649, in run\_until\_complete

return future.result()

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/alphaOpenfoam\_v2.py", line 25, in main

await run\_instance()

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/alphaOpenfoam\_v2.py", line 69, in run\_instance

await env.run()

File "/root/anaconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/environment/base\_env.py", line 167, in run

await asyncio.gather(\*futures)

File "/root/anaconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/utils/common.py", line 584, in wrapper

raise Exception(format\_trackback\_info(limit=None))

Exception: Traceback (most recent call last):

File "/root/anaconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/utils/common.py", line 562, in wrapper

return await func(self, \*args, \*\*kwargs)

File "/root/anaconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/roles/role.py", line 575, in run

rsp = await self.react()

File "/root/anaconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/roles/role.py", line 542, in react

rsp= await self.\_react()

File "/root/anaconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/roles/role.py", line 488, in \_react

rsp = await self.\_act()

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/roles/InputWriter.py", line 36, in \_act

code\_text = await todo.run(context)

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/actions/InputWriterAction.py", line 72, in run

self.save\_file(file\_path, code\_context=str(code\_text))

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/actions/InputWriterAction.py", line 182, in save\_file

with open(file\_path, 'w') as file:

OSError: [Errno 36] File name too long: '/data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/0/U foamfile in 0 folder that could be used to meet user requirement:You are a CFD analyst and need to call the OpenFOAM tutorial to simulate the template example and present the results to the user. Now you have to meet the following requirements: 1. Do a RANS simulation of flow around a cylinder using overPimpleDyMFoam. The simulation investigates the steady flow field around a circular cylinder in an incompressible fluid at Reynolds number of 100. Case name: Cylinder\_Flow\_0. 2. Ensure that the simulation file content is consistent with similar'

### Find case

### actions.ArchitectAction:run:111 - ```splits into 12 subtasks:

## roles.InputWriter, simulate into writting case\_files

### find\_similar\_foamfile, U

### Input U file (X)

## roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

## roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

### review: InputWriter

### review: Runner

## review done, reach max loops 20