# Prompt&input

usr\_requirment: "You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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.  Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks."

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)

### 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

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 expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

./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 23:20:37.762 | INFO | roles.Architect:\_act:26 - Zhuxu: to do ArchitectAction(ArchitectAction)

self.rc.history: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

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

case domain: incompressible

case category: RANS

case solver: simpleFoam

find\_case page\_content="case name: simpleCar\ncase domain: incompressible\ncase category: None\ncase solver: simpleFoam\ncase input name:['U', 'epsilon', 'nut', 'k', 'p', 'blockMeshDict', 'referencePressure', 'fieldAverage', 'createPatchDict', 'runTimeControls', 'controlDict', 'forceCoeffs', 'topoSetDict', 'fvSchemes', 'fvOptions', 'fvSolution', 'turbulenceProperties', 'transportProperties']\ncorresponding input folder:{'U': '0.orig', 'epsilon': '0.orig', 'nut': '0.orig', 'k': '0.orig', 'p': '0.orig', 'blockMeshDict': 'system', 'referencePressure': 'system', 'fieldAverage': 'system', 'createPatchDict': 'system', 'runTimeControls': 'system', 'controlDict': 'system', 'forceCoeffs': 'system', 'topoSetDict': 'system', 'fvSchemes': 'system', 'fvOptions': 'system', 'fvSolution': 'system', '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 23:22:57.375 | 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 expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask2: to Write a OpenFoam epsilon foamfile in 0 folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask3: to Write a OpenFoam nut foamfile in 0 folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask4: to Write a OpenFoam k foamfile in 0 folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask5: to Write a OpenFoam p foamfile in 0 folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask6: to Write a OpenFoam blockMeshDict foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask7: to Write a OpenFoam referencePressure foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask8: to Write a OpenFoam fieldAverage foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask9: to Write a OpenFoam createPatchDict foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask10: to Write a OpenFoam runTimeControls foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask11: to Write a OpenFoam controlDict foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask12: to Write a OpenFoam forceCoeffs foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask13: to Write a OpenFoam topoSetDict foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask14: to Write a OpenFoam fvSchemes foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask15: to Write a OpenFoam fvOptions foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask16: to Write a OpenFoam fvSolution foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask17: to Write a OpenFoam turbulenceProperties foamfile in constant folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

subtask18: to Write a OpenFoam transportProperties foamfile in constant folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

```

2024-08-28 23:22:57.379 | 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 expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam epsilon foamfile in 0 folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam nut foamfile in 0 folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam k foamfile in 0 folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam p foamfile in 0 folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam blockMeshDict foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam referencePressure foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam fieldAverage foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam createPatchDict foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam runTimeControls foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam controlDict foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam forceCoeffs foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam topoSetDict foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam fvSchemes foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam fvOptions foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam fvSolution foamfile in system folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam turbulenceProperties foamfile in constant folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks., user: to Write a OpenFoam transportProperties foamfile in constant folder that could be used to meet user requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.]

tutorial\_file: ```input\_file\_begin: input U file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) 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 volVectorField;

object U;

}

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

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

internalField uniform (10 0 0);

// Surface normal with time ramping

intakeType1

{

type surfaceNormalFixedValue;

refValue uniform 1.2;

ramp table ((0 0) (10 1));

}

// Uniform surface normal with Function1 for ramping

intakeType2

{

type uniformNormalFixedValue;

uniformValue table ((0 0) (10 1.2));

}

// Uniform surface normal with time ramping

intakeType3

{

// Or directly with uniform value (ramping also possible)

type uniformNormalFixedValue;

uniformValue constant 1.2;

ramp table ((0 0) (10 1));

}

boundaryField

{

inlet

{

type fixedValue;

value $internalField;

}

airIntake

{

$intakeType1;

}

outlet

{

type pressureInletOutletVelocity;

value $internalField;

}

"(body|upperWall|lowerWall)"

{

type fixedValue;

value uniform (0 0 0);

}

frontAndBack

{

type empty;

}

}

#remove "intakeType.\*"

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

input\_file\_end.

File U is going to be written

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

folder\_name 0

file\_name U

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/0/U

tutorial\_file: ```input\_file\_begin: input epsilon file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) 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 epsilon;

}

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

dimensions [0 2 -3 0 0 0 0];

internalField uniform 0.125;

boundaryField

{

inlet

{

type fixedValue;

value $internalField;

}

"(body|upperWall|lowerWall)"

{

type epsilonWallFunction;

value $internalField;

Cmu 0.09;

kappa 0.41;

E 9.8;

}

frontAndBack

{

type empty;

}

".\*"

{

type zeroGradient;

}

}

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

input\_file\_end.

File epsilon is going to be written

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

folder\_name 0

file\_name epsilon

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/0/epsilon

tutorial\_file: ```input\_file\_begin: input nut file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) 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 nut;

}

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

dimensions [0 2 -1 0 0 0 0];

internalField uniform 0;

boundaryField

{

"(body|upperWall|lowerWall)"

{

type nutkWallFunction;

Cmu 0.09;

kappa 0.41;

E 9.8;

value $internalField;

}

frontAndBack

{

type empty;

}

".\*"

{

type calculated;

value $internalField;

}

}

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

input\_file\_end.

File nut is going to be written

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

folder\_name 0

file\_name nut

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/0/nut

tutorial\_file: ```input\_file\_begin: input k file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) 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 k;

}

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

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

internalField uniform 0.375;

boundaryField

{

inlet

{

type fixedValue;

value $internalField;

}

"(body|upperWall|lowerWall)"

{

type kqRWallFunction;

value $internalField;

}

frontAndBack

{

type empty;

}

".\*"

{

type zeroGradient;

}

}

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

input\_file\_end.

File k is going to be written

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

folder\_name 0

file\_name k

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/0/k

tutorial\_file: ```input\_file\_begin: input p file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) 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;

}

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

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

internalField uniform 0;

boundaryField

{

outlet

{

type fixedValue;

value $internalField;

}

frontAndBack

{

type empty;

}

".\*"

{

type zeroGradient;

}

}

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

input\_file\_end.

File p is going to be written

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

folder\_name 0

file\_name p

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/0/p

tutorial\_file: ```input\_file\_begin: input blockMeshDict file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in system 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 dictionary;

object blockMeshDict;

}

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

scale 1;

vertices

(

( 0.0 0.0 0.1) // 0

( 2.1 0.0 0.1) // 1

( 2.5 0.0 0.1) // 2

( 3.0 0.0 0.1) // 3

( 5.0 0.0 0.1) // 4

(10.0 0.0 0.1) // 5

( 0.0 0.3 0.1) // 6

( 2.1 0.3 0.1) // 7

( 2.5 0.3 0.1) // 8

( 3.0 0.3 0.1) // 9

( 5.0 0.3 0.1) // 10

(10.0 0.3 0.1) // 11

( 0.0 0.4 0.1) // 12

( 2.0 0.4 0.1) // 13

( 2.4 0.4 0.1) // 14

( 0.0 0.8 0.1) // 15

( 2.0 0.8 0.1) // 16

( 2.6 0.8 0.1) // 17

( 0.0 0.9 0.1) // 18

( 2.1 0.9 0.1) // 19

( 3.0 1.0 0.1) // 20

( 3.6 1.5 0.1) // 21

( 4.4 1.5 0.1) // 22

( 5.0 1.0 0.1) // 23

(10.0 1.0 0.1) // 24

( 0.0 3.0 0.1) // 25

( 1.2 3.0 0.1) // 26

( 2.2 3.0 0.1) // 27

( 3.1 3.0 0.1) // 28

( 4.5 3.0 0.1) // 29

( 5.5 3.0 0.1) // 30

(10.0 3.0 0.1) // 31

( 0.0 0.0 0.0) // 32

( 2.1 0.0 0.0) // 33

( 2.5 0.0 0.0) // 34

( 3.0 0.0 0.0) // 35

( 5.0 0.0 0.0) // 36

(10.0 0.0 0.0) // 37

( 0.0 0.3 0.0) // 38

( 2.1 0.3 0.0) // 39

( 2.5 0.3 0.0) // 40

( 3.0 0.3 0.0) // 41

( 5.0 0.3 0.0) // 42

(10.0 0.3 0.0) // 43

( 0.0 0.4 0.0) // 44

( 2.0 0.4 0.0) // 45

( 2.4 0.4 0.0) // 46

( 0.0 0.8 0.0) // 47

( 2.0 0.8 0.0) // 48

( 2.6 0.8 0.0) // 49

( 0.0 0.9 0.0) // 50

( 2.1 0.9 0.0) // 51

( 3.0 1.0 0.0) // 52

( 3.6 1.5 0.0) // 53

( 4.4 1.5 0.0) // 54

( 5.0 1.0 0.0) // 55

(10.0 1.0 0.0) // 56

( 0.0 3.0 0.0) // 57

( 1.2 3.0 0.0) // 58

( 2.2 3.0 0.0) // 59

( 3.1 3.0 0.0) // 60

( 4.5 3.0 0.0) // 61

( 5.5 3.0 0.0) // 62

(10.0 3.0 0.0) // 63

);

blocks

(

hex ( 0 6 7 1 32 38 39 33) ( 8 40 1) simpleGrading (1 1 1) // 0

hex ( 1 7 8 2 33 39 40 34) ( 8 10 1) simpleGrading (1 1 1) // 1

hex ( 2 8 9 3 34 40 41 35) ( 8 10 1) simpleGrading (1 1 1) // 2

hex ( 3 9 10 4 35 41 42 36) ( 8 40 1) simpleGrading (1 1 1) // 3

hex ( 4 10 11 5 36 42 43 37) ( 8 100 1) simpleGrading (1 1 1) // 4

hex (10 23 24 11 42 55 56 43) ( 15 100 1) simpleGrading (1 1 1) // 5

hex (23 30 31 24 55 62 63 56) ( 40 100 1) simpleGrading (1 1 1) // 6

hex (22 29 30 23 54 61 62 55) ( 40 20 1) simpleGrading (1 1 1) // 7

hex (21 28 29 22 53 60 61 54) ( 40 25 1) simpleGrading (1 1 1) // 8

hex (20 27 28 21 52 59 60 53) ( 40 20 1) simpleGrading (1 1 1) // 9

hex (19 26 27 20 51 58 59 52) ( 40 20 1) simpleGrading (1 1 1) // 10

hex (18 25 26 19 50 57 58 51) ( 40 40 1) simpleGrading (1 1 1) // 11

hex (15 18 19 16 47 50 51 48) ( 3 40 1) simpleGrading (1 1 1) // 12

hex (12 15 16 13 44 47 48 45) ( 10 40 1) simpleGrading (1 1 1) // 13

hex ( 6 12 13 7 38 44 45 39) ( 3 40 1) simpleGrading (1 1 1) // 14

hex (13 16 17 14 45 48 49 46) ( 10 10 1) simpleGrading (1 1 1) // 15

hex (14 17 9 8 46 49 41 40) ( 10 10 1) simpleGrading (1 1 1) // 16

);

edges

(

arc 10 23 (5.15 0.7 0.1)

arc 42 55 (5.15 0.7 0.0)

);

defaultPatch

{

name frontAndBack;

type empty;

}

boundary

(

inlet

{

type patch;

faces

(

( 0 6 38 32)

( 6 12 44 38)

(12 15 47 44)

(15 18 50 47)

(18 25 57 50)

);

}

outlet

{

type patch;

faces

(

(5 37 43 11)

(11 43 56 24)

(24 56 63 31)

);

}

upperWall

{

type wall;

faces

(

(25 26 58 57)

(26 27 59 58)

(27 28 60 59)

(28 29 61 60)

(29 30 62 61)

(30 31 63 62)

);

}

lowerWall

{

type wall;

faces

(

(0 32 33 1)

(1 33 34 2)

(2 34 35 3)

(3 35 36 4)

(4 36 37 5)

);

}

body

{

type wall;

faces

(

(7 8 40 39)

(14 46 40 8)

(13 45 46 14)

(7 39 45 13)

(16 17 49 48)

(17 9 41 49)

(9 10 42 41)

(10 42 55 23)

(22 54 55 23)

(21 53 54 22)

(20 52 53 21)

(19 51 52 20)

(16 48 51 19)

);

}

);

mergePatchPairs

(

);

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

input\_file\_end.

File blockMeshDict is going to be written

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

folder\_name system

file\_name blockMeshDict

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/blockMeshDict

tutorial\_file: ```input\_file\_begin: input referencePressure file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in system folder:

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

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

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

| \\ / O peration | Version: v2312 |

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

| \\/ M anipulation | |

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

sample1

{

type sets;

libs (sampling);

interpolationScheme cellPointFace;

setFormat raw;

fields ( p );

sets

{

cloud

{

type cloud;

axis xyz;

points

(

(1 0.2 0.05)

(1 0.4 0.05)

(1 0.6 0.05)

(1 0.8 0.05)

(1 1.0 0.05)

);

}

}

}

average1

{

type valueAverage;

libs (fieldFunctionObjects);

writeControl writeTime;

functionObject sample1;

fields (average(p));

}

reference1

{

type reference;

libs (fieldFunctionObjects);

writeControl writeTime;

field p;

refValue functionObjectValue;

functionObject average1;

functionObjectResult average(p)Mean;

}

reference2

{

type reference;

libs (fieldFunctionObjects);

writeControl writeTime;

field p;

refValue sample;

position (1 0.2 0.05);

}

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

input\_file\_end.

File referencePressure is going to be written

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

folder\_name system

file\_name referencePressure

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/referencePressure

tutorial\_file: ```input\_file\_begin: input fieldAverage file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in system folder:

fieldAverage1

{

type fieldAverage;

libs (fieldFunctionObjects);

triggerStart 1;

timeStart 500;

controlMode timeOrTrigger;

writeControl writeTime;

fields

(

U

{

base iteration;

mean on;

prime2Mean off;

}

);

}

input\_file\_end.

File fieldAverage is going to be written

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

folder\_name system

file\_name fieldAverage

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/fieldAverage

tutorial\_file: ```input\_file\_begin: input createPatchDict file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in system 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 dictionary;

object createPatchDict;

}

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

pointSync false;

// Patches to create

patches

(

{

// Name of new patch

name airIntake;

// Dictionary to construct new patch from

patchInfo

{

type patch;

}

// How to construct: either from 'patches' or 'set'

constructFrom set;

// If constructFrom = patches : names of patches. Wildcards allowed.

patches ();

// If constructFrom = set : name of faceSet

set airIntake;

}

);

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

input\_file\_end.

File createPatchDict is going to be written

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

folder\_name system

file\_name createPatchDict

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/createPatchDict

tutorial\_file: ```input\_file\_begin: input runTimeControls file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in system folder:

runTimeControl1

{

type runTimeControl;

libs (utilityFunctionObjects);

conditions

{

condition1

{

type average;

functionObject forceCoeffs1;

fields (Cd);

tolerance 1e-3;

window 20;

windowType exact;

}

}

satisfiedAction setTrigger;

trigger 1;

}

runTimeControl2

{

type runTimeControl;

libs (utilityFunctionObjects);

controlMode trigger;

triggerStart 1;

conditions

{

condition1

{

type maxDuration;

duration 100;

}

}

satisfiedAction end;

}

/\*

runTimeControl3

{

type runTimeControl;

libs (utilityFunctionObjects);

conditions

{

condition1

{

type equationInitialResidual;

fields (U.component(0));

value 1e-02;

mode minimum;

}

}

satisfiedAction end;

}

\*/

input\_file\_end.

File runTimeControls is going to be written

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

folder\_name system

file\_name runTimeControls

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/runTimeControls

tutorial\_file: ```input\_file\_begin: input controlDict file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in system 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 dictionary;

object controlDict;

}

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

application simpleFoam;

startFrom startTime;

startTime 0;

stopAt endTime;

endTime 5000;

deltaT 1;

writeControl timeStep;

writeInterval 100;

purgeWrite 10;

writeFormat ascii;

writePrecision 6;

writeCompression off;

timeFormat general;

timePrecision 6;

runTimeModifiable true;

functions

{

#include "forceCoeffs"

#include "fieldAverage"

#include "referencePressure"

#include "runTimeControls"

}

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

input\_file\_end.

File controlDict is going to be written

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

folder\_name system

file\_name controlDict

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/controlDict

tutorial\_file: ```input\_file\_begin: input forceCoeffs file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in system folder:

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

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

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

| \\ / O peration | Version: v2312 |

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

| \\/ M anipulation | |

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

forceCoeffs1

{

type forceCoeffs;

libs (forces);

writeControl writeTime;

writeFields true;

patches (body);

p p;

U U;

rho rhoInf; // Indicates incompressible

log true;

rhoInf 1; // Required when rho = rhoInf

liftDir (0 1 0);

dragDir (1 0 0);

CofR (3.5 0 0); // Axle midpoint on ground

pitchAxis (0 0 1);

magUInf 10;

lRef 4; // Wheelbase length

Aref 1; // Estimated

porosity on;

}

binField1

{

type binField;

libs (fieldFunctionObjects);

binModel singleDirectionUniformBin;

fields (forceCoeff);

patches (body);

decomposePatchValues yes;

CofR ${../forceCoeffs1/CofR};

cellZones (porousZone);

binData

{

nBin 20; // output data into 20 bins

direction (1 0 0); // bin direction

cumulative yes;

}

writeControl writeTime;

}

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

input\_file\_end.

File forceCoeffs is going to be written

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

folder\_name system

file\_name forceCoeffs

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/forceCoeffs

tutorial\_file: ```input\_file\_begin: input topoSetDict file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in system 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 dictionary;

object topoSetDict;

}

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

actions

(

{

name porousCells;

type cellSet;

action new;

source boxToCell;

box (2.05 0.4 -1) (2.1 0.85 1);

}

{

name porousZone;

type cellZoneSet;

action new;

source setToCellZone;

set porousCells;

}

{

name airIntake;

type faceSet;

action new;

source patchToFace;

patch body;

}

{

name airIntake;

type faceSet;

action subset;

source boxToFace;

box (2.6 0.75 0)(2.64 0.8 0.1);

}

);

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

input\_file\_end.

File topoSetDict is going to be written

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

folder\_name system

file\_name topoSetDict

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/topoSetDict

tutorial\_file: ```input\_file\_begin: input fvSchemes file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in system 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 dictionary;

object fvSchemes;

}

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

ddtSchemes

{

default steadyState;

}

gradSchemes

{

default Gauss linear;

}

divSchemes

{

default none;

div(phi,U) bounded Gauss upwind;

turbulence bounded Gauss upwind;

div(phi,k) $turbulence;

div(phi,epsilon) $turbulence;

div((nuEff\*dev2(T(grad(U))))) Gauss linear;

}

laplacianSchemes

{

default Gauss linear corrected;

}

interpolationSchemes

{

default linear;

}

snGradSchemes

{

default corrected;

}

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

input\_file\_end.

File fvSchemes is going to be written

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

folder\_name system

file\_name fvSchemes

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/fvSchemes

tutorial\_file: ```input\_file\_begin: input fvOptions file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in system 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 dictionary;

object fvOptions;

}

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

porosity1

{

type explicitPorositySource;

active true;

explicitPorositySourceCoeffs

{

type DarcyForchheimer;

selectionMode cellZone;

cellZone porousZone;

DarcyForchheimerCoeffs

{

d d [0 -2 0 0 0 0 0] (5e7 -1000 -1000);

f f [0 -1 0 0 0 0 0] (0 0 0);

coordinateSystem

{

origin (0 0 0);

e1 (1 0 0);

e2 (0 1 0);

}

}

}

}

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

input\_file\_end.

File fvOptions is going to be written

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

folder\_name system

file\_name fvOptions

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/fvOptions

tutorial\_file: ```input\_file\_begin: input fvSolution file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in system 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 dictionary;

object fvSolution;

}

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

solvers

{

p

{

solver GAMG;

tolerance 1e-06;

relTol 0.1;

smoother GaussSeidel;

nPreSweeps 0;

nPostSweeps 2;

cacheAgglomeration on;

agglomerator faceAreaPair;

nCellsInCoarsestLevel 10;

mergeLevels 1;

}

"(U|k|epsilon|R|nuTilda)"

{

solver smoothSolver;

smoother symGaussSeidel;

tolerance 1e-05;

relTol 0.1;

}

}

SIMPLE

{

nNonOrthogonalCorrectors 0;

residualControl

{

p 1e-2;

U 1e-4;

"(k|epsilon)" 1e-4;

}

}

relaxationFactors

{

fields

{

p 0.3;

}

equations

{

U 0.7;

k 0.7;

epsilon 0.7;

}

}

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

input\_file\_end.

File fvSolution is going to be written

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

folder\_name system

file\_name fvSolution

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/fvSolution

tutorial\_file: ```input\_file\_begin: input turbulenceProperties file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in constant 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 dictionary;

object turbulenceProperties;

}

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

simulationType RAS;

RAS

{

RASModel kEpsilon;

turbulence on;

printCoeffs on;

}

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

input\_file\_end.

File turbulenceProperties is going to be written

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

folder\_name constant

file\_name turbulenceProperties

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/constant/turbulenceProperties

tutorial\_file: ```input\_file\_begin: input transportProperties file of case simpleCar (domain: incompressible, category: None, solver:simpleFoam) in constant 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 dictionary;

object transportProperties;

}

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

transportModel Newtonian;

nu 1e-05;

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

input\_file\_end.

File transportProperties is going to be written

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

folder\_name constant

file\_name transportProperties

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/constant/transportProperties

2024-08-28 23:28:14.308 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

# 中断

2024-08-28 23:28:24.419 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

2024-08-28 23:28:24.420 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Reviewer: 1

get\_memories\_InputWriter [user: None]

File already exists in . Skipping...

2024-08-28 23:28:24.430 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_write2: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

allrun\_write: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

initial\_files: {'system': {'forceCoeffs', 'fieldAverage', 'createPatchDict', 'blockMeshDict', 'referencePressure', 'fvSolution', 'runTimeControls', 'fvSchemes', 'topoSetDict', 'controlDict', 'fvOptions'}, '0': {'nut', 'p', 'U', 'k', 'epsilon'}, 'constant': {'turbulenceProperties', 'transportProperties'}}

error\_logs: []

No error logs found.

loop: 9

2024-08-28 23:28:24.441 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

2024-08-28 23:28:24.443 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Reviewer: 1

get\_memories\_InputWriter [user: None]

File already exists in . Skipping...

2024-08-28 23:28:24.454 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_write2: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

allrun\_write: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

initial\_files: {'system': {'forceCoeffs', 'fieldAverage', 'createPatchDict', 'blockMeshDict', 'referencePressure', 'fvSolution', 'runTimeControls', 'fvSchemes', 'topoSetDict', 'controlDict', 'fvOptions'}, '0': {'nut', 'p', 'U', 'k', 'epsilon'}, 'constant': {'turbulenceProperties', 'transportProperties'}}

error\_logs: []

No error logs found.

loop: 10

2024-08-28 23:28:24.463 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

2024-08-28 23:28:24.466 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Reviewer: 1

get\_memories\_InputWriter [user: None]

File already exists in . Skipping...

2024-08-28 23:28:24.476 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_write2: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

allrun\_write: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

initial\_files: {'system': {'forceCoeffs', 'fieldAverage', 'createPatchDict', 'blockMeshDict', 'referencePressure', 'fvSolution', 'runTimeControls', 'fvSchemes', 'topoSetDict', 'controlDict', 'fvOptions'}, '0': {'nut', 'p', 'U', 'k', 'epsilon'}, 'constant': {'turbulenceProperties', 'transportProperties'}}

error\_logs: []

No error logs found.

loop: 11

2024-08-28 23:28:24.486 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

2024-08-28 23:28:24.488 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Reviewer: 1

get\_memories\_InputWriter [user: None]

File already exists in . Skipping...

2024-08-28 23:28:24.498 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_write2: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

allrun\_write: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

initial\_files: {'system': {'forceCoeffs', 'fieldAverage', 'createPatchDict', 'blockMeshDict', 'referencePressure', 'fvSolution', 'runTimeControls', 'fvSchemes', 'topoSetDict', 'controlDict', 'fvOptions'}, '0': {'nut', 'p', 'U', 'k', 'epsilon'}, 'constant': {'turbulenceProperties', 'transportProperties'}}

error\_logs: []

No error logs found.

loop: 12

2024-08-28 23:28:24.509 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

2024-08-28 23:28:24.511 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Reviewer: 1

get\_memories\_InputWriter [user: None]

File already exists in . Skipping...

2024-08-28 23:28:24.521 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_write2: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

allrun\_write: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

initial\_files: {'system': {'forceCoeffs', 'fieldAverage', 'createPatchDict', 'blockMeshDict', 'referencePressure', 'fvSolution', 'runTimeControls', 'fvSchemes', 'topoSetDict', 'controlDict', 'fvOptions'}, '0': {'nut', 'p', 'U', 'k', 'epsilon'}, 'constant': {'turbulenceProperties', 'transportProperties'}}

error\_logs: []

No error logs found.

loop: 13

2024-08-28 23:28:24.530 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

2024-08-28 23:28:24.532 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Reviewer: 1

get\_memories\_InputWriter [user: None]

File already exists in . Skipping...

2024-08-28 23:28:24.542 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_write2: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

allrun\_write: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

initial\_files: {'system': {'forceCoeffs', 'fieldAverage', 'createPatchDict', 'blockMeshDict', 'referencePressure', 'fvSolution', 'runTimeControls', 'fvSchemes', 'topoSetDict', 'controlDict', 'fvOptions'}, '0': {'nut', 'p', 'U', 'k', 'epsilon'}, 'constant': {'turbulenceProperties', 'transportProperties'}}

error\_logs: []

No error logs found.

loop: 14

2024-08-28 23:28:24.552 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

2024-08-28 23:28:24.554 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Reviewer: 1

get\_memories\_InputWriter [user: None]

File already exists in . Skipping...

2024-08-28 23:28:24.564 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_write2: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

allrun\_write: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

initial\_files: {'system': {'forceCoeffs', 'fieldAverage', 'createPatchDict', 'blockMeshDict', 'referencePressure', 'fvSolution', 'runTimeControls', 'fvSchemes', 'topoSetDict', 'controlDict', 'fvOptions'}, '0': {'nut', 'p', 'U', 'k', 'epsilon'}, 'constant': {'turbulenceProperties', 'transportProperties'}}

error\_logs: []

No error logs found.

loop: 15

2024-08-28 23:28:24.575 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

2024-08-28 23:28:24.577 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Reviewer: 1

get\_memories\_InputWriter [user: None]

File already exists in . Skipping...

2024-08-28 23:28:24.587 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_write2: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

allrun\_write: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

initial\_files: {'system': {'forceCoeffs', 'fieldAverage', 'createPatchDict', 'blockMeshDict', 'referencePressure', 'fvSolution', 'runTimeControls', 'fvSchemes', 'topoSetDict', 'controlDict', 'fvOptions'}, '0': {'nut', 'p', 'U', 'k', 'epsilon'}, 'constant': {'turbulenceProperties', 'transportProperties'}}

error\_logs: []

No error logs found.

loop: 16

2024-08-28 23:28:24.600 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

2024-08-28 23:28:24.603 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Reviewer: 1

get\_memories\_InputWriter [user: None]

File already exists in . Skipping...

2024-08-28 23:28:24.613 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_write2: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

allrun\_write: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

initial\_files: {'system': {'forceCoeffs', 'fieldAverage', 'createPatchDict', 'blockMeshDict', 'referencePressure', 'fvSolution', 'runTimeControls', 'fvSchemes', 'topoSetDict', 'controlDict', 'fvOptions'}, '0': {'nut', 'p', 'U', 'k', 'epsilon'}, 'constant': {'turbulenceProperties', 'transportProperties'}}

error\_logs: []

No error logs found.

loop: 17

2024-08-28 23:28:24.623 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

2024-08-28 23:28:24.625 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Reviewer: 1

get\_memories\_InputWriter [user: None]

File already exists in . Skipping...

2024-08-28 23:28:24.635 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_write2: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

allrun\_write: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

initial\_files: {'system': {'forceCoeffs', 'fieldAverage', 'createPatchDict', 'blockMeshDict', 'referencePressure', 'fvSolution', 'runTimeControls', 'fvSchemes', 'topoSetDict', 'controlDict', 'fvOptions'}, '0': {'nut', 'p', 'U', 'k', 'epsilon'}, 'constant': {'turbulenceProperties', 'transportProperties'}}

error\_logs: []

No error logs found.

loop: 18

2024-08-28 23:28:24.645 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

2024-08-28 23:28:24.647 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Reviewer: 1

get\_memories\_InputWriter [user: None]

File already exists in . Skipping...

2024-08-28 23:28:24.658 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_write2: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

allrun\_write: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

initial\_files: {'system': {'forceCoeffs', 'fieldAverage', 'createPatchDict', 'blockMeshDict', 'referencePressure', 'fvSolution', 'runTimeControls', 'fvSchemes', 'topoSetDict', 'controlDict', 'fvOptions'}, '0': {'nut', 'p', 'U', 'k', 'epsilon'}, 'constant': {'turbulenceProperties', 'transportProperties'}}

error\_logs: []

No error logs found.

loop: 19

2024-08-28 23:28:24.669 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.

2024-08-28 23:28:24.671 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Reviewer: 1

get\_memories\_InputWriter [user: None]

File already exists in . Skipping...

2024-08-28 23:28:24.681 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_write2: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

allrun\_write: #!/bin/sh

cd "${0%/\*}" || exit # Run from this directory

. ${WM\_PROJECT\_DIR:?}/bin/tools/RunFunctions # Tutorial run functions

#------------------------------------------------------------------------------

# Copy necessary files

cp -f \

"$FOAM\_TUTORIALS"/resources/geometry/cylinder.stl.gz \

constant/triSurface/

# Prepare the case directory

mv 0.orig 0

restore0Dir

# Mesh generation and modification

runApplication blockMesh

runApplication surfaceFeatureExtract

runApplication snappyHexMesh -overwrite

# Run the simulation

runApplication $(getApplication)

# Post-processing

runApplication reconstructParMesh -constant

runApplication reconstructPar

#------------------------------------------------------------------------------

initial\_files: {'system': {'forceCoeffs', 'fieldAverage', 'createPatchDict', 'blockMeshDict', 'referencePressure', 'fvSolution', 'runTimeControls', 'fvSchemes', 'topoSetDict', 'controlDict', 'fvOptions'}, '0': {'nut', 'p', 'U', 'k', 'epsilon'}, 'constant': {'turbulenceProperties', 'transportProperties'}}

error\_logs: []

No error logs found.

loop: 20

reach max loops 20

Subdirectory File Count Line Count

======================================================================

system 11 466

0 5 212

constant 2 35

======================================================================

Total 18 713

2024-08-28 23:28:24.691 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

files\_names: ['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'p', 'k', 'U', 'turbulenceProperties', 'transportProperties'] {'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'p': '0', 'k': '0', 'U': '0', 'turbulenceProperties': 'constant', 'transportProperties': 'constant'}

command: None

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of flow around a cylinder using simpleFoam. 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. Also 1. Copy the content after the sample is found. 2. There is no word empty. 3.0.orig folder changed to 0. 4. File name without quotation marks.