# 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 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.  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 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. 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 23:40:18.600 | 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 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. 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: cylinder

case solver: overPimpleDyMFoam

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

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

# 中断

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

2024-08-28 23:58:27.586 | 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

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

Deleted file: /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/blockMesh.err

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

log\_file: log.snappyHexMesh

log\_file: log.simpleFoam

log\_file: log.surfaceFeatureExtract

log\_file: log.reconstructPar

log\_file: log.blockMesh

log\_file: log.reconstructParMesh

error\_logs: [{'file': 'log.snappyHexMesh', 'error\_content': '/\*---------------------------------------------------------------------------\*\\\n| ========= | |\n| \\\\ / F ield | OpenFOAM: The Open Source CFD Toolbox |\n| \\\\ / O peration | Version: 2312 |\n| \\\\ / A nd | Website: www.openfoam.com |\n| \\\\/ M anipulation | |\n\\\*---------------------------------------------------------------------------\*/\nBuild : \_e651d635-20240208 OPENFOAM=2312 patch=240220 version=2312\nArch : "LSB;label=32;scalar=64"\nExec : snappyHexMesh -overwrite\nDate : Aug 28 2024\nTime : 23:58:28\nHost : ubuntu\nPID : 36081\nI/O : uncollated\nCase : /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1\nnProcs : 1\ntrapFpe: Floating point exception trapping enabled (FOAM\_SIGFPE).\nfileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)\nallowSystemOperations : Allowing user-supplied system call operations\n\n// \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* //\nCreate time\n\nCreate mesh for time = 0\n\n\n--> FOAM FATAL ERROR: (openfoam-2312 patch=240220)\nCannot find file "points" in directory "polyMesh" in times "0" down to constant\n\nFrom virtual Foam::IOobject Foam::fileOperation::findInstance(const Foam::IOobject&, Foam::scalar, const Foam::word&) const\nin file global/fileOperations/fileOperation/fileOperation.C at line 1211.\n\nFOAM exiting\n'}, {'file': 'log.simpleFoam', 'error\_content': '/\*---------------------------------------------------------------------------\*\\\n| ========= | |\n| \\\\ / F ield | OpenFOAM: The Open Source CFD Toolbox |\n| \\\\ / O peration | Version: 2312 |\n| \\\\ / A nd | Website: www.openfoam.com |\n| \\\\/ M anipulation | |\n\\\*---------------------------------------------------------------------------\*/\nBuild : \_e651d635-20240208 OPENFOAM=2312 patch=240220 version=2312\nArch : "LSB;label=32;scalar=64"\nExec : simpleFoam\nDate : Aug 28 2024\nTime : 23:58:28\nHost : ubuntu\nPID : 36085\nI/O : uncollated\nCase : /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1\nnProcs : 1\ntrapFpe: Floating point exception trapping enabled (FOAM\_SIGFPE).\nfileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)\nallowSystemOperations : Allowing user-supplied system call operations\n\n// \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* //\nCreate time\n\nCreate mesh for time = 0\n\n\n--> FOAM FATAL ERROR: (openfoam-2312 patch=240220)\nCannot find file "points" in directory "polyMesh" in times "0" down to constant\n\nFrom virtual Foam::IOobject Foam::fileOperation::findInstance(const Foam::IOobject&, Foam::scalar, const Foam::word&) const\nin file global/fileOperations/fileOperation/fileOperation.C at line 1211.\n\nFOAM exiting\n'}, {'file': 'log.surfaceFeatureExtract', 'error\_content': '| ========= | |\n| \\\\ / F ield | OpenFOAM: The Open Source CFD Toolbox |\n| \\\\ / O peration | Version: 2312 |\n| \\\\ / A nd | Website: www.openfoam.com |\n| \\\\/ M anipulation | |\n\\\*---------------------------------------------------------------------------\*/\nBuild : \_e651d635-20240208 OPENFOAM=2312 patch=240220 version=2312\nArch : "LSB;label=32;scalar=64"\nExec : surfaceFeatureExtract\nDate : Aug 28 2024\nTime : 23:58:27\nHost : ubuntu\nPID : 36080\nI/O : uncollated\nCase : /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1\nnProcs : 1\ntrapFpe: Floating point exception trapping enabled (FOAM\_SIGFPE).\nfileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)\nallowSystemOperations : Allowing user-supplied system call operations\n\n// \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* //\nCreate time\n\n\nNote: Feature line extraction only valid on closed manifold surfaces\n\nReading surfaceFeatureExtractDict\n\n\n\n--> FOAM FATAL ERROR: (openfoam-2312 patch=240220)\ncannot find file "/data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/surfaceFeatureExtractDict"\n\nFrom virtual Foam::autoPtr<Foam::ISstream> Foam::fileOperations::uncollatedFileOperation::readStream(Foam::regIOobject&, const Foam::fileName&, const Foam::word&, bool) const\nin file global/fileOperations/uncollatedFileOperation/uncollatedFileOperation.C at line 626.\n\nFOAM exiting\n'}, {'file': 'log.reconstructPar', 'error\_content': '/\*---------------------------------------------------------------------------\*\\\n| ========= | |\n| \\\\ / F ield | OpenFOAM: The Open Source CFD Toolbox |\n| \\\\ / O peration | Version: 2312 |\n| \\\\ / A nd | Website: www.openfoam.com |\n| \\\\/ M anipulation | |\n\\\*---------------------------------------------------------------------------\*/\nBuild : \_e651d635-20240208 OPENFOAM=2312 patch=240220 version=2312\nArch : "LSB;label=32;scalar=64"\nExec : reconstructPar\nDate : Aug 28 2024\nTime : 23:58:28\nHost : ubuntu\nPID : 36087\nI/O : uncollated\nCase : /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1\nnProcs : 1\ntrapFpe: Floating point exception trapping enabled (FOAM\_SIGFPE).\nfileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)\nallowSystemOperations : Allowing user-supplied system call operations\n\n// \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* //\nCreate time\n\n\n\n--> FOAM FATAL ERROR: (openfoam-2312 patch=240220)\nNo processor\* directories found\n\nFrom int main(int, char\*\*)\nin file reconstructPar.C at line 227.\n\nFOAM exiting\n'}, {'file': 'log.blockMesh', 'error\_content': '/\*---------------------------------------------------------------------------\*\\\n| ========= | |\n| \\\\ / F ield | OpenFOAM: The Open Source CFD Toolbox |\n| \\\\ / O peration | Version: 2312 |\n| \\\\ / A nd | Website: www.openfoam.com |\n| \\\\/ M anipulation | |\n\\\*---------------------------------------------------------------------------\*/\nBuild : \_e651d635-20240208 OPENFOAM=2312 patch=240220 version=2312\nArch : "LSB;label=32;scalar=64"\nExec : blockMesh\nDate : Aug 28 2024\nTime : 23:58:27\nHost : ubuntu\nPID : 35969\nI/O : uncollated\nCase : /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1\nnProcs : 1\ntrapFpe: Floating point exception trapping enabled (FOAM\_SIGFPE).\nfileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)\nallowSystemOperations : Allowing user-supplied system call operations\n\n// \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* //\nCreate time\n\nCreating block mesh from "system/blockMeshDict"\nCreating block edges\n\n\n--> FOAM FATAL ERROR: (openfoam-2312 patch=240220)\n0\n\nFrom void Foam::blockEdges::arcEdge::calcFromMidPoint(const point&, const point&, const point&)\nin file blockEdges/arcEdge/arcEdge.C at line 66.\n\nFOAM aborting\n\n[stack trace]\n=============\n#1 Foam::error::simpleExit(int, bool) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libOpenFOAM.so\n#2 Foam::error::exiting(int, bool) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libOpenFOAM.so\n#3 Foam::blockEdges::arcEdge::calcFromMidPoint(Foam::Vector<double> const&, Foam::Vector<double> const&, Foam::Vector<double> const&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so\n#4 Foam::blockEdges::arcEdge::arcEdge(Foam::dictionary const&, int, Foam::searchableSurfaces const&, Foam::Field<Foam::Vector<double> > const&, Foam::Istream&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so\n#5 Foam::blockEdge::addIstreamConstructorToTable<Foam::blockEdges::arcEdge>::New(Foam::dictionary const&, int, Foam::searchableSurfaces const&, Foam::Field<Foam::Vector<double> > const&, Foam::Istream&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so\n#6 Foam::blockEdge::New(Foam::dictionary const&, int, Foam::searchableSurfaces const&, Foam::Field<Foam::Vector<double> > const&, Foam::Istream&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so\n#7 void Foam::PtrList<Foam::blockEdge>::readIstream<Foam::blockEdge::iNew>(Foam::Istream&, Foam::blockEdge::iNew const&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so\n#8 Foam::blockMesh::createTopology(Foam::IOdictionary const&, Foam::word const&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so\n#9 Foam::blockMesh::blockMesh(Foam::IOdictionary const&, Foam::word const&, Foam::blockMesh::mergeStrategy, int) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so\n#10 ? in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/bin/blockMesh\n#11 \_\_libc\_start\_main in /usr/lib/x86\_64-linux-gnu/libc.so.6\n#12 ? in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/bin/blockMesh\n============='}, {'file': 'log.reconstructParMesh', 'error\_content': 'Build : \_e651d635-20240208 OPENFOAM=2312 patch=240220 version=2312\nArch : "LSB;label=32;scalar=64"\nExec : reconstructParMesh -constant\nDate : Aug 28 2024\nTime : 23:58:28\nHost : ubuntu\nPID : 36086\nI/O : uncollated\nCase : /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1\nnProcs : 1\ntrapFpe: Floating point exception trapping enabled (FOAM\_SIGFPE).\nfileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)\nallowSystemOperations : Allowing user-supplied system call operations\n\n// \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* \* //\nCreate time\n\nMerge individual processor meshes back into one master mesh.\nUse if the original master mesh has been deleted or the processor meshes\nhave been modified (topology change).\nThis tool will write the resulting mesh to a new time step and construct\nxxxxProcAddressing files in the processor meshes so reconstructPar can be\nused to regenerate the fields on the master mesh.\n\nNot well tested & use at your own risk!\n\nMerge assuming correct, fully matched procBoundaries.\n\n\n\n--> FOAM FATAL ERROR: (openfoam-2312 patch=240220)\nNo processor\* directories found\n\nFrom int main(int, char\*\*)\nin file reconstructParMesh.C at line 711.\n\nFOAM exiting\n'}]

command line error: blockMesh

error: /\*---------------------------------------------------------------------------\*\

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

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

| \\ / O peration | Version: 2312 |

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

| \\/ M anipulation | |

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

Build : \_e651d635-20240208 OPENFOAM=2312 patch=240220 version=2312

Arch : "LSB;label=32;scalar=64"

Exec : blockMesh

Date : Aug 28 2024

Time : 23:58:27

Host : ubuntu

PID : 35969

I/O : uncollated

Case : /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1

nProcs : 1

trapFpe: Floating point exception trapping enabled (FOAM\_SIGFPE).

fileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)

allowSystemOperations : Allowing user-supplied system call operations

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

Create time

Creating block mesh from "system/blockMeshDict"

Creating block edges

--> FOAM FATAL ERROR: (openfoam-2312 patch=240220)

0

From void Foam::blockEdges::arcEdge::calcFromMidPoint(const point&, const point&, const point&)

in file blockEdges/arcEdge/arcEdge.C at line 66.

FOAM aborting

[stack trace]

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

#1 Foam::error::simpleExit(int, bool) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libOpenFOAM.so

#2 Foam::error::exiting(int, bool) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libOpenFOAM.so

#3 Foam::blockEdges::arcEdge::calcFromMidPoint(Foam::Vector<double> const&, Foam::Vector<double> const&, Foam::Vector<double> const&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#4 Foam::blockEdges::arcEdge::arcEdge(Foam::dictionary const&, int, Foam::searchableSurfaces const&, Foam::Field<Foam::Vector<double> > const&, Foam::Istream&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#5 Foam::blockEdge::addIstreamConstructorToTable<Foam::blockEdges::arcEdge>::New(Foam::dictionary const&, int, Foam::searchableSurfaces const&, Foam::Field<Foam::Vector<double> > const&, Foam::Istream&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#6 Foam::blockEdge::New(Foam::dictionary const&, int, Foam::searchableSurfaces const&, Foam::Field<Foam::Vector<double> > const&, Foam::Istream&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#7 void Foam::PtrList<Foam::blockEdge>::readIstream<Foam::blockEdge::iNew>(Foam::Istream&, Foam::blockEdge::iNew const&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#8 Foam::blockMesh::createTopology(Foam::IOdictionary const&, Foam::word const&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#9 Foam::blockMesh::blockMesh(Foam::IOdictionary const&, Foam::word const&, Foam::blockMesh::mergeStrategy, int) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#10 ? in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/bin/blockMesh

#11 \_\_libc\_start\_main in /usr/lib/x86\_64-linux-gnu/libc.so.6

#12 ? in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/bin/blockMesh

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

final\_files: {'system': {'setFieldsDict', 'fvOptions', 'controlDict', 'createPatchDict', 'topoSetDict', 'fvSolution', 'forceCoeffs', 'fieldAverage', 'blockMeshDict', 'fvSchemes', 'decomposeParDict', 'referencePressure', 'runTimeControls'}, '0': {'U', 'p', 'zoneID', 'cellDisplacement', 'pointDisplacement', 'epsilon', 'nut', 'k'}, 'constant': {'dynamicMeshDict', 'turbulenceProperties', 'transportProperties', 'thermophysicalProperties'}}

loop: 20

reach max loops 20

Subdirectory File Count Line Count

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

system 13 528

0 8 326

constant 4 95

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

Total 25 949

2024-08-28 23:58:28.328 | INFO | roles.Reviewer:\_act:22 - Xingyu: to do ReviewerAction(ReviewerAction)

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

command: blockMesh

requirement: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: 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. 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.

prompt3

blockMesh has been executed in openfoam10, and got the following error:

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

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

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

| \\ / O peration | Version: 2312 |

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

| \\/ M anipulation | |

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

Build : \_e651d635-20240208 OPENFOAM=2312 patch=240220 version=2312

Arch : "LSB;label=32;scalar=64"

Exec : blockMesh

Date : Aug 28 2024

Time : 23:58:27

Host : ubuntu

PID : 35969

I/O : uncollated

Case : /data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1

nProcs : 1

trapFpe: Floating point exception trapping enabled (FOAM\_SIGFPE).

fileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 5, maxFileModificationPolls 20)

allowSystemOperations : Allowing user-supplied system call operations

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

Create time

Creating block mesh from "system/blockMeshDict"

Creating block edges

--> FOAM FATAL ERROR: (openfoam-2312 patch=240220)

0

From void Foam::blockEdges::arcEdge::calcFromMidPoint(const point&, const point&, const point&)

in file blockEdges/arcEdge/arcEdge.C at line 66.

FOAM aborting

[stack trace]

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

#1 Foam::error::simpleExit(int, bool) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libOpenFOAM.so

#2 Foam::error::exiting(int, bool) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libOpenFOAM.so

#3 Foam::blockEdges::arcEdge::calcFromMidPoint(Foam::Vector<double> const&, Foam::Vector<double> const&, Foam::Vector<double> const&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#4 Foam::blockEdges::arcEdge::arcEdge(Foam::dictionary const&, int, Foam::searchableSurfaces const&, Foam::Field<Foam::Vector<double> > const&, Foam::Istream&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#5 Foam::blockEdge::addIstreamConstructorToTable<Foam::blockEdges::arcEdge>::New(Foam::dictionary const&, int, Foam::searchableSurfaces const&, Foam::Field<Foam::Vector<double> > const&, Foam::Istream&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#6 Foam::blockEdge::New(Foam::dictionary const&, int, Foam::searchableSurfaces const&, Foam::Field<Foam::Vector<double> > const&, Foam::Istream&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#7 void Foam::PtrList<Foam::blockEdge>::readIstream<Foam::blockEdge::iNew>(Foam::Istream&, Foam::blockEdge::iNew const&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#8 Foam::blockMesh::createTopology(Foam::IOdictionary const&, Foam::word const&) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#9 Foam::blockMesh::blockMesh(Foam::IOdictionary const&, Foam::word const&, Foam::blockMesh::mergeStrategy, int) in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/lib/libblockMesh.so

#10 ? in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/bin/blockMesh

#11 \_\_libc\_start\_main in /usr/lib/x86\_64-linux-gnu/libc.so.6

#12 ? in /usr/lib/openfoam/openfoam2312/platforms/linux64GccDPInt32Opt/bin/blockMesh

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

The corresponding input file list is:

['referencePressure', 'controlDict', 'createPatchDict', 'topoSetDict', 'setFieldsDict', 'fieldAverage', 'fvOptions', 'blockMeshDict', 'fvSolution', 'decomposeParDict', 'runTimeControls', 'fvSchemes', 'forceCoeffs', 'epsilon', 'nut', 'pointDisplacement', 'p', 'cellDisplacement', 'k', 'zoneID', 'U', 'dynamicMeshDict', 'turbulenceProperties', 'thermophysicalProperties', 'transportProperties']

The corresponding directories are:

{'referencePressure': 'system', 'controlDict': 'system', 'createPatchDict': 'system', 'topoSetDict': 'system', 'setFieldsDict': 'system', 'fieldAverage': 'system', 'fvOptions': 'system', 'blockMeshDict': 'system', 'fvSolution': 'system', 'decomposeParDict': 'system', 'runTimeControls': 'system', 'fvSchemes': 'system', 'forceCoeffs': 'system', 'epsilon': '0', 'nut': '0', 'pointDisplacement': '0', 'p': '0', 'cellDisplacement': '0', 'k': '0', 'zoneID': '0', 'U': '0', 'dynamicMeshDict': 'constant', 'turbulenceProperties': 'constant', 'thermophysicalProperties': 'constant', 'transportProperties': 'constant'}

Please analyze whether the error is related to the file structure, such as missing critical files or redundant input files.

If the error is related to the file structure, return the updated file structure list and their respective directories in the following format:

###file\_name1, file\_name2, ...### in ```file\_folder1, file\_folder2, ...``` with NO other texts.

If the error is not related to file structure, return:

``` None ```

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

rsp\_structrue: ``` None ```

# Ai solution

从提供的日志来看，运行 OpenFOAM 的过程中出现了多个严重错误，导致仿真无法正常进行。以下是问题的总结和可能的解决方案：

* **1. OpenFOAM 版本问题**

日志显示，当前使用的 OpenFOAM 版本为 2312。如果你希望使用 OpenFOAM 10，那么需要确保脚本和环境变量正确指向 OpenFOAM 10。

* **2. 错误 1: arcEdge 错误**

plaintext

复制代码

FOAM FATAL ERROR: (openfoam-2312 patch=240220)

0

From void Foam::blockEdges::arcEdge::calcFromMidPoint(const point&, const point&, const point&)

in file blockEdges/arcEdge/arcEdge.C at line 66.

**问题分析**: 这个错误与 blockMeshDict 中定义的 arc 有关。可能是因为中点、起点、终点的定义有问题，导致无法正确生成弧形边。

**解决方案**: 重新检查并修正 blockMeshDict 文件中 arc 的定义，确保中点的坐标在起点和终点之间，并且符合弧的生成要求。

* **3. 错误 2: 无法找到 processor\* 目录**

plaintext

复制代码

FOAM FATAL ERROR: (openfoam-2312 patch=240220)

No processor\* directories found

**问题分析**: 这个错误发生在 reconstructPar 和 reconstructParMesh 命令执行时，通常表明你尝试进行并行计算的结果重构，但没有相应的并行计算结果目录。

**解决方案**: 如果没有使用并行计算，请不要调用 reconstructPar 和 reconstructParMesh。如果使用了并行计算，请检查是否正确生成了 processor\* 目录。

* **4. 错误 3: 无法找到 points 文件**

plaintext

复制代码

FOAM FATAL ERROR: (openfoam-2312 patch=240220)

Cannot find file "points" in directory "polyMesh" in times "0" down to constant

**问题分析**: 这个错误通常表明网格未成功生成，导致无法找到 points 文件。

**解决方案**: 检查网格生成步骤（如 blockMesh 或 snappyHexMesh）是否成功执行。如果没有成功，需要先解决这些步骤中的问题。

* **5. 错误 4: 无法找到 surfaceFeatureExtractDict 文件**

plaintext

复制代码

FOAM FATAL ERROR: (openfoam-2312 patch=240220)

cannot find file "/data/sda/lichenshuo/XiMuAlpha4CFD/run/Cylinder\_Flow\_0\_1/system/surfaceFeatureExtractDict"

**问题分析**: surfaceFeatureExtract 命令依赖 surfaceFeatureExtractDict 配置文件，但该文件不存在或路径错误。

**解决方案**: 确保 system 目录下存在 surfaceFeatureExtractDict 文件，并且配置正确。如果该文件缺失，需要手动创建或从类似的案例中复制。

* **6. 建议措施**
* **环境变量设置**: 确保脚本和环境指向正确的 OpenFOAM 版本（如 OpenFOAM 10）。可以通过 source /opt/openfoam10/etc/bashrc 设置环境变量。
* **文件和目录检查**: 检查所有必需的文件和目录是否存在并且路径正确。如果缺少某些文件或目录，可以从其他案例复制或手动创建。
* **重新运行关键步骤**: 在解决文件和路径问题后，重新运行 blockMesh、snappyHexMesh 和其他生成网格和初始化的步骤，确保所有步骤都成功完成。
* **总结**

日志显示在运行 OpenFOAM 时存在多个错误，涉及到版本问题、文件路径、以及网格生成和处理过程中的问题。建议首先确保使用正确的 OpenFOAM 版本，检查并修复文件和目录的路径问题，然后逐步重新执行网格生成和仿真步骤，确保每一步都成功完成。如果需要使用 OpenFOAM 10，请确保环境变量和脚本都指向 OpenFOAM 10。