# Prompt&input

usr\_requirment: "You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of buoyantCavity using buoyantSimpleFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829.  Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder 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

# 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

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

Please select an input file from the list below:

1) BuoyantCavity\_0.yaml 5) Cavity\_RANS.yaml 9) HIT.yaml

2) BuoyantCavity\_0829.yaml 6) Combustion.yaml 10) PitzDaily.yaml

3) BuoyantCavity\_pre.yaml 7) CylinderFlow.yaml 11) Planar\_Poiseuille.yaml

4) Cavity.yaml 8) CylinderFlow\_0.yaml 12) SquareBendLiq.yaml

#? 2

You selected BuoyantCavity\_0829.yaml

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

config\_file\_path inputs/BuoyantCavity\_0829.yaml

Configuration loaded successfully:

usr\_requirment: You are a CFD expert, call OpenFOAM sample simulation. The requirements are: do a RANS simulation of buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

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

runtimes: 1

/root/miniconda3/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(

2024-08-29 09:38:07.345 | 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

user\_case: Standard format:

case name: Buoyant\_Cavity\_0829\_2

case domain: heatTransfer

case category: None

case solver: buoyantFoam

find\_case page\_content="case name: buoyantCavity\ncase domain: heatTransfer\ncase category: None\ncase solver: buoyantFoam\ncase input name:['U', 'omega', 'T', 'alphat', 'epsilon', 'nut', 'p\_rgh', 'k', 'p', 'createGraphs', 'blockMeshDict', 'sample', 'controlDict', 'fvSchemes', 'fvSolution', 'pRef', 'physicalProperties', 'momentumTransport', 'g']\ncorresponding input folder:{'U': '0', 'omega': '0', 'T': '0', 'alphat': '0', 'epsilon': '0', 'nut': '0', 'p\_rgh': '0', 'k': '0', 'p': '0', 'createGraphs': 'validation', 'blockMeshDict': 'system', 'sample': 'system', 'controlDict': 'system', 'fvSchemes': 'system', 'fvSolution': 'system', 'pRef': 'constant', 'physicalProperties': 'constant', 'momentumTransport': 'constant', 'g': 'constant'}" metadata={'source': '/data/sda/lichenshuo/XiMuAlpha4CFD/database/openfoam\_tutorials\_summary.txt'}

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Buoyant\_Cavity\_0829\_2\_1/find\_tutorial.txt

2024-08-29 09:41:30.815 | INFO | actions.ArchitectAction:run:111 - ```splits into 19 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask2: to Write a OpenFoam omega 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask3: to Write a OpenFoam T 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask4: to Write a OpenFoam alphat 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask5: 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask6: 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask7: to Write a OpenFoam p\_rgh 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask8: 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask9: 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask10: to Write a OpenFoam createGraphs foamfile in validation 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask11: 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask12: to Write a OpenFoam sample 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask13: 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask15: 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask16: to Write a OpenFoam pRef 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask17: to Write a OpenFoam physicalProperties 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask18: to Write a OpenFoam momentumTransport 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

subtask19: to Write a OpenFoam g 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.

```

2024-08-29 09:41:30.822 | INFO | roles.InputWriter:\_act:24 - Yuxuan: to do InputWriterAction(InputWriterAction)

number\_subtasks Architect: 19

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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks., user: to Write a OpenFoam omega 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks., user: to Write a OpenFoam T 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks., user: to Write a OpenFoam alphat 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks., user: to Write a OpenFoam p\_rgh 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks., user: to Write a OpenFoam createGraphs foamfile in validation 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks., user: to Write a OpenFoam sample 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks., user: to Write a OpenFoam pRef 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks., user: to Write a OpenFoam physicalProperties 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks., user: to Write a OpenFoam momentumTransport 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks., user: to Write a OpenFoam g 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 buoyantCavity using buoyantFoam, which investigate natural convection in a heat cavity with a temperature difference of 20K is maintained between the hot and cold; the remaining patches are treated as adiabatic, case name: Buoyant\_Cavity\_0829\_2. Also 1. Copy the content after the sample is found. 2. No word empty in any file. 3.Copy 0.orig folder to 0. 4. File name without quotation marks.]

tutorial\_file: ```input\_file\_begin: input U file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in 0 folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class volVectorField;

location "0";

object U;

}

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

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

internalField uniform (0 0 0);

boundaryField

{

frontAndBack

{

type noSlip;

}

topAndBottom

{

type noSlip;

}

hot

{

type noSlip;

}

cold

{

type noSlip;

}

}

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

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/Buoyant\_Cavity\_0829\_2\_1/0/U

tutorial\_file: ```input\_file\_begin: input omega file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in 0 folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class volScalarField;

location "0";

object omega;

}

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

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

internalField uniform 0.12;

boundaryField

{

frontAndBack

{

type omegaWallFunction;

value uniform 0.12;

}

topAndBottom

{

type omegaWallFunction;

value uniform 0.12;

}

hot

{

type omegaWallFunction;

value uniform 0.12;

}

cold

{

type omegaWallFunction;

value uniform 0.12;

}

}

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

input\_file\_end.

File omega is going to be written

tutorial\_file: ```input\_file\_begin: input p\_rgh file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in 0 folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class volScalarField;

location "0";

object p\_rgh;

}

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

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

internalField uniform 0;

boundaryField

{

frontAndBack

{

type fixedFluxPressure;

value $internalField;

}

topAndBottom

{

type fixedFluxPressure;

value $internalField;

}

hot

{

type fixedFluxPressure;

value $internalField;

}

cold

{

type fixedFluxPressure;

value $internalField;

}

}

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

input\_file\_end.

File p\_rgh is going to be written

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

folder\_name 0

file\_name p\_rgh

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Buoyant\_Cavity\_0829\_2\_1/0/p\_rgh

tutorial\_file: ```input\_file\_begin: input k file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in 0 folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class volScalarField;

location "0";

object k;

}

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

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

internalField uniform 3.75e-04;

boundaryField

{

frontAndBack

{

type kqRWallFunction;

value uniform 3.75e-04;

}

topAndBottom

{

type kqRWallFunction;

value uniform 3.75e-04;

}

hot

{

type kqRWallFunction;

value uniform 3.75e-04;

}

cold

{

type kqRWallFunction;

value uniform 3.75e-04;

}

}

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

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/Buoyant\_Cavity\_0829\_2\_1/0/k

tutorial\_file: ```input\_file\_begin: input p file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in 0 folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class volScalarField;

location "0";

object p;

}

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

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

internalField uniform 1e5;

boundaryField

{

frontAndBack

{

type calculated;

value $internalField;

}

topAndBottom

{

type calculated;

value $internalField;

}

hot

{

type calculated;

value $internalField;

}

cold

{

type calculated;

value $internalField;

}

}

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

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/Buoyant\_Cavity\_0829\_2\_1/0/p

tutorial\_file: ```input\_file\_begin: input createGraphs file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in validation folder:

#!/bin/sh

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

# ========= |

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

# \\ / O peration | Website: https://openfoam.org

# \\ / A nd | Copyright (C) 2011-2022 OpenFOAM Foundation

# \\/ M anipulation |

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

# License

# This file is part of OpenFOAM.

#

# OpenFOAM is free software: you can redistribute it and/or modify it

# under the terms of the GNU General Public License as published by

# the Free Software Foundation, either version 3 of the License, or

# (at your option) any later version.

#

# OpenFOAM is distributed in the hope that it will be useful, but WITHOUT

# ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or

# FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License

# for more details.

#

# You should have received a copy of the GNU General Public License

# along with OpenFOAM. If not, see <http://www.gnu.org/licenses/>.

#

# Script

# createGraphs

#

# Description

# Creates .eps graphs of OpenFOAM results vs experiment for the buoyant

# cavity case

#

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

cd ${0%/\*} || exit 1 # Run from this directory

# Stop on first error

set -e

createEpsT()

{

index=$1

OF=$2

EXPT=$3

gnuplot<<EOF

set terminal postscript eps color enhanced

set output "OF\_vs\_EXPT\_T$i.eps"

set xlabel "Channel width, x / [m]"

set ylabel "Temperature / [K]"

set grid

set key left top

set size 0.6, 0.6

set xrange [0:0.08]

set yrange [285:310]

plot \

"$EXPT" u (\$1/1000):(\$2+273.15) title "Expt 0.$index" \

with points lt 1 pt 6, \

"$OF" title "OpenFOAM 0.$index" with lines linetype -1

EOF

}

createEpsU()

{

index=$1

OF=$2

EXPT=$3

gnuplot<<EOF

set terminal postscript eps color enhanced

set output "OF\_vs\_EXPT\_U$i.eps"

set xlabel "Channel width, x / [m]"

set ylabel "Vertical velocity component, Uy / [m/s]"

set grid

set key left top

set size 0.6, 0.6

set xrange [0:0.08]

set yrange [-0.2:0.2]

plot \

"$EXPT" u (\$1/1000):(\$2) title "Expt 0.$index" \

with points lt 1 pt 6, \

"$OF" u 1:4 title "OpenFOAM 0.$index" with lines linetype -1

EOF

}

echo "createGraphs:"

# test if gnuplot exists on the system

if ! which gnuplot > /dev/null 2>&1

then

echo "FOAM FATAL ERROR: gnuplot not found - skipping graph creation" >&2

exit 1

fi

SETSDIR="../postProcessing/sample"

if [ ! -d $SETSDIR ]

then

echo "FOAM FATAL ERROR: result sets not available in directory $SETSDIR" >&2

exit 1

fi

# paths to data

LATESTTIME=`ls $SETSDIR`

OFDATAROOT=$SETSDIR/$LATESTTIME

EXPTDATAROOT=./exptData

# generate temperature profiles

TSets="1 3 4 5 6 7 9"

for i in $TSets

do

echo " processing temperature profile at y/yMax of 0.$i"

OF="$OFDATAROOT/y0.${i}.xy"

EXPT="$EXPTDATAROOT/mt\_z0\_${i}0\_lo.dat"

createEpsT $i $OF $EXPT

done

# generate velocity profiles

USets="1 3 4 5 6 7 9"

for i in $USets

do

echo " processing velocity profile at y/yMax of 0.$i"

OF="$OFDATAROOT/y0.${i}.xy"

EXPT="$EXPTDATAROOT/mv\_z0\_${i}0\_lo.dat"

createEpsU $i $OF $EXPT

done

echo "End"

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

input\_file\_end.

File createGraphs is going to be written

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

folder\_name validation

file\_name createGraphs

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Buoyant\_Cavity\_0829\_2\_1/validation/createGraphs

tutorial\_file: ```input\_file\_begin: input blockMeshDict file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in system folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class dictionary;

object blockMeshDict;

}

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

convertToMeters 0.001;

vertices

(

( 0 0 -260)

(76 0 -260)

(76 2180 -260)

( 0 2180 -260)

( 0 0 260)

(76 0 260)

(76 2180 260)

( 0 2180 260)

);

blocks

(

hex (0 1 2 3 4 5 6 7) (35 150 15) simpleGrading (1 1 1)

);

boundary

(

topAndBottom

{

type wall;

faces

(

(0 1 5 4)

(2 3 7 6)

);

}

frontAndBack

{

type wall;

faces

(

(4 5 6 7)

(3 2 1 0)

);

}

hot

{

type wall;

faces

(

(6 5 1 2)

);

}

cold

{

type wall;

faces

(

(4 7 3 0)

);

}

);

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

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/Buoyant\_Cavity\_0829\_2\_1/system/blockMeshDict

tutorial\_file: ```input\_file\_begin: input sample file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in system folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class dictionary;

location "system";

object sample;

}

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

type sets;

libs ("libsampling.so");

interpolationScheme cellPointFace;

setFormat raw;

sets

(

y0.1

{

type lineFace;

axis x;

start (-1 0.218 0);

end (1 0.218 0);

}

y0.2

{

type lineFace;

axis x;

start (-1 0.436 0);

end (1 0.436 0);

}

y0.3

{

type lineFace;

axis x;

start (-1 0.654 0);

end (1 0.654 0);

}

y0.4

{

type lineFace;

axis x;

start (-1 0.872 0);

end (1 0.872 0);

}

y0.5

{

type lineFace;

axis x;

start (-1 1.09 0);

end (1 1.09 0);

}

y0.6

{

type lineFace;

axis x;

start (-1 1.308 0);

end (1 1.308 0);

}

y0.7

{

type lineFace;

axis x;

start (-1 1.526 0);

end (1 1.526 0);

}

y0.8

{

type lineFace;

axis x;

start (-1 1.744 0);

end (1 1.744 0);

}

y0.9

{

type lineFace;

axis x;

start (-1 1.962 0);

end (1 1.962 0);

}

);

fields

(

T

U

);

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

input\_file\_end.

File sample is going to be written

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

folder\_name system

file\_name sample

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Buoyant\_Cavity\_0829\_2\_1/system/sample

tutorial\_file: ```input\_file\_begin: input controlDict file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in system folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class dictionary;

object controlDict;

}

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

application buoyantFoam;

startFrom startTime;

startTime 0;

stopAt endTime;

endTime 1000;

deltaT 1;

writeControl timeStep;

writeInterval 50;

purgeWrite 3;

writeFormat ascii;

writePrecision 6;

writeCompression off;

timeFormat general;

timePrecision 6;

runTimeModifiable true;

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

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/Buoyant\_Cavity\_0829\_2\_1/system/controlDict

tutorial\_file: ```input\_file\_begin: input fvSchemes file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in system folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class dictionary;

object fvSchemes;

}

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

ddtSchemes

{

default steadyState;

}

gradSchemes

{

default Gauss linear;

}

divSchemes

{

default none;

div(phi,U) bounded Gauss limitedLinear 0.2;

div(phi,K) bounded Gauss limitedLinear 0.2;

div(phi,h) bounded Gauss limitedLinear 0.2;

div(phi,k) bounded Gauss limitedLinear 0.2;

div(phi,epsilon) bounded Gauss limitedLinear 0.2;

div(phi,omega) bounded Gauss limitedLinear 0.2;

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

}

laplacianSchemes

{

default Gauss linear orthogonal;

}

interpolationSchemes

{

default linear;

}

snGradSchemes

{

default orthogonal;

}

wallDist

{

method meshWave;

}

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

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/Buoyant\_Cavity\_0829\_2\_1/system/fvSchemes

tutorial\_file: ```input\_file\_begin: input fvSolution file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in system folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class dictionary;

location "system";

object fvSolution;

}

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

solvers

{

p\_rgh

{

solver GAMG;

tolerance 1e-7;

relTol 0.01;

smoother DICGaussSeidel;

}

"(U|h|k|epsilon|omega)"

{

solver PBiCGStab;

preconditioner DILU;

tolerance 1e-8;

relTol 0.1;

}

}

PIMPLE

{

momentumPredictor no;

nNonOrthogonalCorrectors 0;

pRefCell 0;

pRefValue 0;

residualControl

{

p\_rgh 1e-4;

U 1e-4;

h 1e-4;

// possibly check turbulence fields

"(k|epsilon|omega)" 1e-3;

}

}

relaxationFactors

{

fields

{

rho 1.0;

p\_rgh 0.7;

}

equations

{

U 0.3;

h 0.3;

"(k|epsilon|omega)" 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/Buoyant\_Cavity\_0829\_2\_1/system/fvSolution

tutorial\_file: ```input\_file\_begin: input pRef file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in constant folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class uniformDimensionedScalarField;

location "constant";

object pRef;

}

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

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

value 1e5;

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

input\_file\_end.

File pRef is going to be written

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

folder\_name constant

file\_name pRef

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Buoyant\_Cavity\_0829\_2\_1/constant/pRef

tutorial\_file: ```input\_file\_begin: input physicalProperties file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in constant folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class dictionary;

location "constant";

object physicalProperties;

}

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

thermoType

{

type heRhoThermo;

mixture pureMixture;

transport const;

thermo hConst;

equationOfState perfectGas;

specie specie;

energy sensibleEnthalpy;

}

mixture

{

specie

{

molWeight 28.96;

}

thermodynamics

{

Cp 1004.4;

Hf 0;

}

transport

{

mu 1.831e-05;

Pr 0.705;

}

}

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

input\_file\_end.

File physicalProperties is going to be written

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

folder\_name constant

file\_name physicalProperties

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Buoyant\_Cavity\_0829\_2\_1/constant/physicalProperties

tutorial\_file: ```input\_file\_begin: input momentumTransport file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in constant folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class dictionary;

object RASProperties;

}

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

simulationType RAS;

RAS

{

model kOmegaSST;

turbulence on;

printCoeffs on;

}

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

input\_file\_end.

File momentumTransport is going to be written

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

folder\_name constant

file\_name momentumTransport

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Buoyant\_Cavity\_0829\_2\_1/constant/momentumTransport

tutorial\_file: ```input\_file\_begin: input g file of case buoyantCavity (domain: heatTransfer, category: None, solver:buoyantFoam) in constant folder:

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

========= |

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

\\ / O peration | Website: https://openfoam.org

\\ / A nd | Version: 10

\\/ M anipulation |

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

FoamFile

{

format ascii;

class uniformDimensionedVectorField;

location "constant";

object g;

}

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

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

value (0 -9.81 0);

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

input\_file\_end.

File g is going to be written

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

folder\_name constant

file\_name g

File saved successfully at /data/sda/lichenshuo/XiMuAlpha4CFD/run/Buoyant\_Cavity\_0829\_2\_1/constant/g

2024-08-29 09:49:01.215 | INFO | roles.Runner:\_act:20 - Foamer: to do RunnerAction(RunnerAction)

allrun\_total: ```sh

#!/bin/sh

cd ${0%/\*} || exit 1 # Run from this directory

# Source tutorial run functions

. $WM\_PROJECT\_DIR/bin/tools/RunFunctions

# Copy 0.orig folder to 0

cp -r 0.orig 0

# Set application name

application=$(getApplication)

runApplication blockMesh

runApplication $application

runApplication -s sample postProcess -latestTime -func sample

runApplication validation/createGraphs

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

```

allrun\_write: #!/bin/sh

cd ${0%/\*} || exit 1 # Run from this directory

# Source tutorial run functions

. $WM\_PROJECT\_DIR/bin/tools/RunFunctions

# Copy 0.orig folder to 0

cp -r 0.orig 0

# Set application name

application=$(getApplication)

runApplication blockMesh

runApplication $application

runApplication -s sample postProcess -latestTime -func sample

runApplication validation/createGraphs

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

initial\_files: {'system': {'fvSchemes', 'sample', 'blockMeshDict', 'controlDict', 'fvSolution'}, 'validation': {'createGraphs'}, '0': {'epsilon', 'T', 'U', 'k', 'alphat', 'nut', 'p', 'omega', 'p\_rgh'}, 'constant': {'physicalProperties', 'pRef', 'momentumTransport', 'g'}}

log\_file: log.

log\_file: log.createGraphs

log\_file: log.postProcess.sample

log\_file: log.blockMesh

error\_logs: []

2024-08-29 09:49:08.352 | WARNING | alpha.utils.common:wrapper:571 - There is a exception in role's execution, in order to resume, we delete the newest role communication message in the role's memory.

Traceback (most recent call last):

File "/root/miniconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/utils/common.py", line 562, in wrapper

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

File "/root/miniconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/roles/role.py", line 575, in run

rsp = await self.react()

File "/root/miniconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/roles/role.py", line 542, in react

rsp= await self.\_react()

File "/root/miniconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/roles/role.py", line 488, in \_react

rsp = await self.\_act()

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/roles/Runner.py", line 23, in \_act

msg = await todo.run(context)

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/actions/RunnerAction.py", line 110, in run

commands\_run = self.extract\_commands\_from\_allrun\_out(out\_file)

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/actions/RunnerAction.py", line 313, in extract\_commands\_from\_allrun\_out

command\_true = command.split()[0]

IndexError: list index out of range

During handling of the above exception, another exception occurred:

Traceback (most recent call last):

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

asyncio.run(main())

File "/root/miniconda3/envs/ximualpha/lib/python3.10/asyncio/runners.py", line 44, in run

return loop.run\_until\_complete(main)

File "/root/miniconda3/envs/ximualpha/lib/python3.10/asyncio/base\_events.py", line 649, in run\_until\_complete

return future.result()

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

await run\_instance()

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

await env.run()

File "/root/miniconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/environment/base\_env.py", line 167, in run

await asyncio.gather(\*futures)

File "/root/miniconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/utils/common.py", line 584, in wrapper

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

Exception: Traceback (most recent call last):

File "/root/miniconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/utils/common.py", line 562, in wrapper

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

File "/root/miniconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/roles/role.py", line 575, in run

rsp = await self.react()

File "/root/miniconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/roles/role.py", line 542, in react

rsp= await self.\_react()

File "/root/miniconda3/envs/ximualpha/lib/python3.10/site-packages/alpha/roles/role.py", line 488, in \_react

rsp = await self.\_act()

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/roles/Runner.py", line 23, in \_act

msg = await todo.run(context)

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/actions/RunnerAction.py", line 110, in run

commands\_run = self.extract\_commands\_from\_allrun\_out(out\_file)

File "/data/sda/lichenshuo/XiMuAlpha4CFD/src/actions/RunnerAction.py", line 313, in extract\_commands\_from\_allrun\_out

command\_true = command.split()[0]

IndexError: list index out of range

# Ai solution

* **您说：**