# 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

# Ai solution

* **您说：**

这是算例4的需求、输出文件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" cp: cannot stat '0.orig': No such file or directory Running blockMesh on /data/sda/lichenshuo/XiMuAlpha4CFD/run/Buoyant\_Cavity\_0829\_1 Running buoyantSimpleFoam on /data/sda/lichenshuo/XiMuAlpha4CFD/run/Buoyant\_Cavity\_0829\_1 Running postProcess on /data/sda/lichenshuo/XiMuAlpha4CFD/run/Buoyant\_Cavity\_0829\_1 Running validation/createGraphs on /data/sda/lichenshuo/XiMuAlpha4CFD/run/Buoyant\_Cavity\_0829\_1 blockMesh: /usr/lib/x86\_64-linux-gnu/libc.so.6: version GLIBC\_2.34' not found (required by blockMesh) blockMesh: /usr/lib/x86\_64-linux-gnu/libm.so.6: version GLIBC\_2.35' not found (required by /opt/openfoam10/platforms/linux64GccDPInt32Opt/lib/libOpenFOAM.so) blockMesh: /usr/lib/x86\_64-linux-gnu/libc.so.6: version GLIBC\_2.33' not found (required by /opt/openfoam10/platforms/linux64GccDPInt32Opt/lib/libOpenFOAM.so) blockMesh: /usr/lib/x86\_64-linux-gnu/libc.so.6: version GLIBC\_2.34' not found (required by /opt/openfoam10/platforms/linux64GccDPInt32Opt/lib/libOpenFOAM.so) The buoyantSimpleFoam solver has been replaced by the more general buoyantFoam solver which can be used for both steady and transient cases. /opt/openfoam10/bin/tools/RunFunctions: line 93: validation/createGraphs: No such file or directory postProcess: /usr/lib/x86\_64-linux-gnu/libc.so.6: version GLIBC\_2.34' not found (required by postProcess) postProcess: /usr/lib/x86\_64-linux-gnu/libm.so.6: version GLIBC\_2.35' not found (required by /opt/openfoam10/platforms/linux64GccDPInt32Opt/lib/libOpenFOAM.so) postProcess: /usr/lib/x86\_64-linux-gnu/libc.so.6: version GLIBC\_2.33' not found (required by /opt/openfoam10/platforms/linux64GccDPInt32Opt/lib/libOpenFOAM.so) postProcess: /usr/lib/x86\_64-linux-gnu/libc.so.6: version GLIBC\_2.34' not found (required by /opt/openfoam10/platforms/linux64GccDPInt32Opt/lib/libOpenFOAM.so)

* **ChatGPT 说：**

ChatGPT

从你提供的日志来看，执行 OpenFOAM 仿真任务时遇到了多个关键问题。以下是问题的详细分析和建议的解决方案：

* **1. 缺少 0.orig 目录**

plaintext

复制代码

cp: cannot stat '0.orig': No such file or directory

**问题分析**: 这个错误表明在尝试复制 0.orig 目录时，系统找不到该目录。这个目录通常包含初始条件文件。

**解决方案**:

* 确保在 OpenFOAM 仿真案例的路径下存在 0.orig 目录。
* 如果没有 0.orig 目录，你可能需要手动创建并填充所需的初始条件文件。
* **2. GLIBC 版本不匹配**

plaintext

复制代码

blockMesh: /usr/lib/x86\_64-linux-gnu/libc.so.6: version `GLIBC\_2.34' not found (required by blockMesh)

**问题分析**: 这个错误表明你的系统上的 GLIBC 版本太低，而 OpenFOAM 需要更高版本的 GLIBC 来运行。

**解决方案**:

* **升级 GLIBC**: 这通常比较复杂且具有风险，可能需要手动编译和安装更高版本的 GLIBC。
* **使用 Docker**: 如果升级 GLIBC 不可行，可以考虑使用带有合适环境的 Docker 容器来运行 OpenFOAM。
* **3. 求解器问题**

plaintext

复制代码

The buoyantSimpleFoam solver has been replaced by the more general buoyantFoam solver

**问题分析**: OpenFOAM 中的 buoyantSimpleFoam 求解器已经被弃用，替代为更通用的 buoyantFoam。

**解决方案**:

* 更新你的配置文件或脚本，使用 buoyantFoam 代替 buoyantSimpleFoam。
* 确保所有相关设置和文件都适配新的求解器。
* **4. 后处理和验证问题**

plaintext

复制代码

/opt/openfoam10/bin/tools/RunFunctions: line 93: validation/createGraphs: No such file or directory

**问题分析**: 脚本试图运行 validation/createGraphs，但找不到这个文件或目录。

**解决方案**:

* 确保 validation/createGraphs 文件或目录存在并且路径正确。
* 如果此步骤不是必须的，可以暂时忽略，或者修改脚本以跳过这部分。
* **5. 总结**

你当前遇到的主要问题是系统的 GLIBC 版本与 OpenFOAM 所需的不匹配。解决这个问题最直接的方法是：

1. **升级 GLIBC**: 但这可能会引入其他兼容性问题，不建议轻易尝试。
2. **使用 Docker**: 通过 Docker 可以使用官方的 OpenFOAM 镜像，这些镜像已经预装了正确版本的依赖库，避免了版本兼容性问题。

另外，确保使用 buoyantFoam 作为求解器，并检查所有必需的文件和目录是否存在，可以帮助你成功运行仿真。