1. $FOAM TUTORIALS/incompressible/icoFoam/cavity/cavity效果很好？
2. $FOAM\_TUTORIALS/heatTransfer/buoyantSimpleFoam/buoyantCavity会出错主要是以下问题：

文件夹加单引号

P文件指令empty

Allrun文件求解器设置有误buoyantSimpleFoam设置为buoyantFoam

这一步没问题，然而创建文件出错——完全复制不更改可以保证不错。

###case begin:

case name: buoyantCavity

case domain: heatTransfer

case category: None

case solver: buoyantSimpleFoam

case input name:['U', 'omega', 'T', 'alphat', 'epsilon', 'nut', 'p\_rgh', 'k', 'p', 'plot', 'blockMeshDict', 'sample', 'controlDict', 'fvSchemes', 'fvSolution', 'turbulenceProperties', 'thermophysicalProperties', 'g']

corresponding input folder:{'U': '0.orig', 'omega': '0.orig', 'T': '0.orig', 'alphat': '0.orig', 'epsilon': '0.orig', 'nut': '0.orig', 'p\_rgh': '0.orig', 'k': '0.orig', 'p': '0.orig', 'plot': 'validation', 'blockMeshDict': 'system', 'sample': 'system', 'controlDict': 'system', 'fvSchemes': 'system', 'fvSolution': 'system', 'turbulenceProperties': 'constant', 'thermophysicalProperties': 'constant', 'g': 'constant'}

case end.###