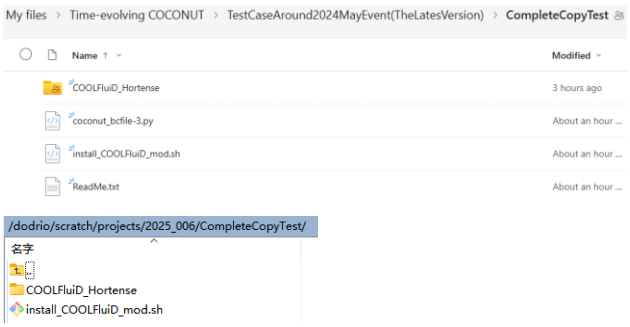
Time-evolving COCONUT installation

1. Download the folder “CompleteCopyTest” from the following link and upload them to your HPC cluster section.

   TestCaseAround2024MayEvent(TheLatesVersion)



2. Modify the path definition of “TOP_DIR” and “COOLFLUID_TOP_DIR” in the script “install_COOLFluiD_mod.sh” according to your case.

   For my case, it is:

   export TOP_DIR="${VSC_SCRATCH_PROJECTS_BASE}/2025_006/CompleteCopyTest/"

   export COOLFLUID_TOP_DIR="${TOP_DIR}/COOLFluiD_Hortense/"

3. Goto the directory containing “install_COOLFluiD_mod.sh” and perform a full recompile:

   cd /dodrio/scratch/projects/2025_006/ CompleteCopyTest/

   chmod a+x ./install_COOLFluiD_mod.sh

   cd ./COOLFluiD_Hortense

   chmod a+x ./prepare.pl

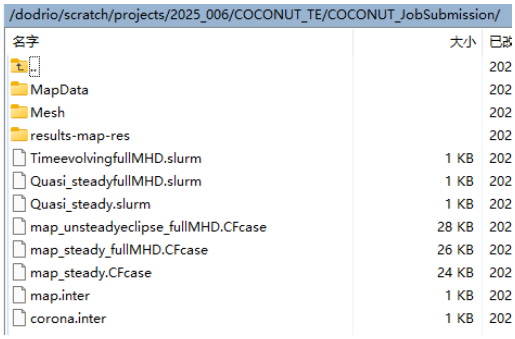
   cd ../

   ./install_COOLFluiD_mod.sh DEBUG NOCUDA –download=2

   This command won’t replace the uploaded files but ensure a successful recompile.

Run time-evolving COCONUT simulation

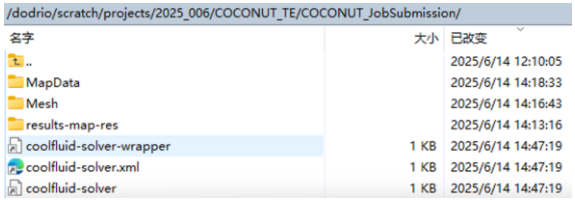
①Download the files shared in OneDrive and included in “../TestCaseAround2024MayEvent(TheLatesVersion)/Scratch_DiskSection/COCONUT_Map” and then upload them to your working folder, for my case, they are uploaded as follows.



②Create a symbolic link to the path of the working folder

cd /dodrio/scratch/projects/2025_006/COCONUT_TE/COCONUT_JobSubmission/

ln -sf /dodrio/scratch/projects/2025_006/CompleteCopyTest/COOLFluiD_Hortense/OPENMPI/optim/apps/Solver/coolfluid-solver ./*



③Following the three steps described in the time-evolving COCONUT Manu.

2.3.1 step 1

First, you can open the file named **map_steady.CFcase** and change the path of your data accordingly. The first part is in **Boundary conditions**. The correct one will be like this:

Simulator.SubSystem.EM.Data.DistanceBased.FileNameTw = ./MapData/map_adapt_l-max15_20070910140000.dat The time of this magnetogram is the first data in your studying carrington rotation, corresponding to t_0.

Then you change the file path of the Mesh reader part. Attention you have to mute the restart part. Simulator.SubSystem.CFmeshFileReader0.Data.FileName = ./30Rs_lvl5.CFmesh. the same for Simulator.SubSystem.CFmeshFileReader1.Data.FileName = ./30Rs_lvl5.CFmesh.

Another line to change is in the very bottom of the file and like this:

Simulator.SubSystem.Flow.Data.DistanceBased.FileNameTw = ./MapData/map_adapt_l-max15_20070910140000.dat

Then you will finish the running for step 1 by submitting **Quasi_steady.slurm**. Remember to change the path in this file accordingly.

2.3.2 step 2

After finishing step one, there will be a result file named **corona.CFmesh**. Now you open the file named **map_steady_fullMHD.CFcase** and change the lines of both first reader and second reader like this:

Simulator.SubSystem.CFmeshFileReader0.Data.FileName = ./result/corona.CFmesh. The same for Simulator.SubSystem.CFmeshFileReader1.Data.FileName

Attention this time you will need the Restart part and mute the lines in the step 1.

Again there is another line to change which is in the very bottom of the file and looks like this:

Simulator.SubSystem.Flow.Data.DistanceBased.FileNameTw = ./MapData/map_adapt_l-max15_20070910140000.dat. The same for Simulator.SubSystem.EM.Data.DistanceBased.FileNameTw

Now you can submit the **Quasi_steady_fullMhd.slurm**. Remember to change the path in this file accordingly.

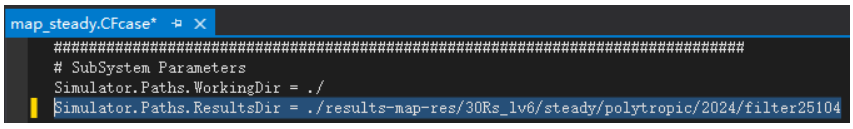
2.3.3 step 3

Now you can open the file named map_unsteadyeclipse_fullMHD.CFcase and change the

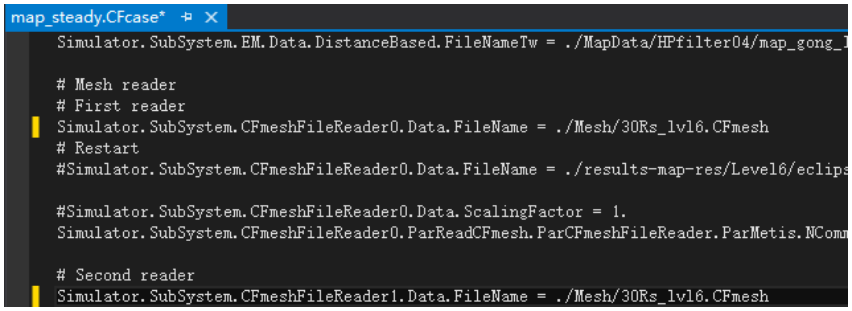
The default is a simulation on Level5 mesh. Here I show how to change it to level 6 mesh.

Step1: Polytropic quasi-steady-state simulation: corresponding to “map_steady.CFcase”

Change the directory where results are stored.



Change to Level 6 mesh



Allocate 540 CPUs for faster calculations and modify the corresponding path

run_Quasi_steady_Tier1.pbs

```
#!/bin/bash -l
#PBS -N group_gpr_compute_2025_006
#PBS -l nodes=6:ppn=90 -l mem=220gb
#PBS -l walltime=1:00:00
#PBS -A 2025_006
##PBS -m abe
##PBS -M haopeng.wang1@kuleuven.be
#PBS -m n


cd /dodrio/scratch/projects/2025_006/COCONUT_TE/COCONUT_JobSubmission/
##module swap cluster/dodrio/cpu_rome_512
module load CMake/3.26.3-GCCcore-12.3.0
module load PETSc/3.20.3-foss-2023a
module load vsc-mympirun


mympirun --universe 540 ./coolfluid-solver --scase /dodrio/scratch/projects/2025_006/COCONUT_TE/COCONUT_JobSubmission/map_steady.CFcase
```

Submit the polytropic simulation via:

qsub run_Quasi_steady_Tier1.pbs

Some log messages from the quasi-steady-state simulation:

```
/dodrio/scratch/projects/2025_006/COCONUT_TE/COCONUT_JobSubmission/group gpr_compute_2025_006.o12033880 - vsc35794@tier1.hpc.ugent.be - 编辑器 - WinSCP
SuperInletProjectionConstrained::preProcess() => END
KSP convergence reached at iteration: 37
[ FlowSubSystemStatus      ] Iter: 88      Res: [    -0.76208067 -0.88840447 -0.86519687 -0.90049164 -1.1484708 -1.2527307 -1.2119609 -1.6322535 -1.0315632 ]  CFL: 2

SuperInletProjectionConstrained::preProcess() => START
SuperInletProjectionConstrained::preProcess() => END
KSP convergence reached at iteration: 37
[ FlowSubSystemStatus      ] Iter: 89      Res: [    -0.76283173 -0.88865108 -0.8656451 -0.90055966 -1.1512543 -1.2544211 -1.2143467 -1.6330581 -1.0382488 ]  CFL: 2

SuperInletProjectionConstrained::preProcess() => START
SuperInletProjectionConstrained::preProcess() => END
KSP convergence reached at iteration: 37
[ FlowSubSystemStatus      ] Iter: 90      Res: [    -0.76357262 -0.88889648 -0.86608965 -0.90062543 -1.1540196 -1.2560942 -1.2167206 -1.6338516 -1.0449439 ]  CFL: 2

SuperInletProjectionConstrained::preProcess() => START
SuperInletProjectionConstrained::preProcess() => END
KSP convergence reached at iteration: 37
```

Step2: FullMHD quasi-steady-state simulation: “map_steady_fullMHD.CFcase”

Change the directory where results are stored, and the corresponding input file calculated in step 1.

map_steady_fullMHD.CFcase

```
################################################################################
# SubSystem Parameters
Simulator.Paths.WorkingDir = ./
Simulator.Paths.ResultsDir = ./results-map-res/30Rs_lv6/steady/fullMHD/2024/filter25104
#Time-dependent
#Simulator.SubSystem.InteractiveParamReader.FileName = ./map.inter
#Simulator.SubSystem.InteractiveParamReader.readRate = 1

# Boundary condition files, another definition for flow field is around Line 461
Simulator.SubSystem.EM.Data.DistanceBased.FileNameTw = ./MapData/HPfilter04/map_gong_lmax25_20240409050400.dat

# Mesh reader
# First reader
#Simulator.SubSystem.CFmeshFileReader0.Data.FileName = ./30Rs_lv16.CFmesh
# Restart
Simulator.SubSystem.CFmeshFileReader0.Data.FileName = ./results-map-res/30Rs_lv6/steady/polytropic/2024/filter25104/corona_720_PP.CFmesh

#Simulator.SubSystem.CFmeshFileReader0.Data.ScalingFactor = 1.
Simulator.SubSystem.CFmeshFileReader0.ParReadCFmesh.ParCFmeshFileReader.ParMetis.NCommonNodes = 2 # ??

# Second reader
#Simulator.SubSystem.CFmeshFileReader1.Data.FileName = ./30Rs_lv16.CFmesh
# Restart
Simulator.SubSystem.CFmeshFileReader1.Data.FileName = ./results-map-res/30Rs_lv6/steady/polytropic/2024/filter25104/corona_720_PP.CFmesh
```

Allocate 540 CPUs for faster calculations and modify the corresponding path

run_Quasi_steadyfullMHD_Tier1.pbs

```
#!/bin/bash -l
#PBS -N group_gpr_compute_2025_006
#PBS -l nodes=6:ppn=90 -l mem=230gb
#PBS -l walltime=2:00:00
#PBS -A 2025_006
##PBS -m abe
##PBS -M haopeng.wang1@kuleuven.be
#PBS -m n


cd /dodrio/scratch/projects/2025_006/COCONUT_TE/COCONUT_JobSubmission/
##module swap cluster/dodrio/cpu_rome_512
module load CMake/3.26.3-GCCcore-12.3.0
module load PETSc/3.20.3-foss-2023a
module load vsc-mympirun


mympirun --universe 540 ./coolfluid-solver --scase /dodrio/scratch/projects/2025_006/COCONUT_TE/COCONUT_JobSubmission/map_steady_fullMHD.CFcase
```

Step3: FullMHD time-evolving simulation: “map_unsteadyeclipse_fullMHD.CFcase”

Change the directory where results are stored, and the corresponding input file calculated in step 2.

map_unsteadyecli...e_fullMHD.CFcase

```
# SubSystem Parameters
Simulator.Paths.WorkingDir = ./
Simulator.Paths.ResultsDir = ./results-map-res/30Rs_lv6/Timeevolving/fullMHD/2024/filter25104
#Time-dependent
Simulator.SubSystem.InteractiveParamReader.FileName = ./map.inter
Simulator.SubSystem.InteractiveParamReader.readRate = 10

# Boundary condition files, another definition for flow field is around Line 466
Simulator.SubSystem.EM.Data.DistanceBased.FileNameTw = ./MapData/HPfilter04/map_gong_lmax25_20240409050400.dat

# Mesh reader
# First reader
#Simulator.SubSystem.CFmeshFileReader0.Data.FileName = ./30Rs_lvl6.CFmesh
# Restart at initial time of time-evolving simulation
Simulator.SubSystem.CFmeshFileReader0.Data.FileName = ./results-map-res/30Rs_lv6/steady/fullMHD/2024/filter25104/corona_720.CFmesh
# Xuguan-Restart during time-dependent simulation: Correspondingly, the following code should be revised.
# L222  CFreal restarttime = 0.0;
# L223  return (PhysicalModelStack::getActive()->getImplementor()->getRefTime())*m_currentTime+restarttime*1447.2/3600.0;
# in \src\Framework\SubSystemStatus.cxx(216):CFreal SubSystemStatus::getCurrentTimeDim() const
#Simulator.SubSystem.CFmeshFileReader0.Data.FileName = ./results-map-res/30Rs_lv6/Timeevolving/fullMHD/2024/filter25104/LayoutandCFme

#Simulator.SubSystem.CFmeshFileReader0.Data.ScalingFactor = 1.
Simulator.SubSystem.CFmeshFileReader0.ParReadCFmesh.ParCFmeshFileReader.ParMetis.NCommonNodes = 2 # ??

# Second reader
#Simulator.SubSystem.CFmeshFileReader1.Data.FileName = ./30Rs_lvl6.CFmesh
# Restart at initial time of time-evolving simulation
Simulator.SubSystem.CFmeshFileReader1.Data.FileName = ./results-map-res/30Rs_lv6/steady/fullMHD/2024/filter25104/corona_720.CFmesh
```

Allocate 288 CPUs but adopt more nodes so that each CPU has more memory storage during the simulation.

run_TimeevolvingfullMHD_Tier1.pbs

```
#!/bin/bash -l
#PBS -N group gpr_compute_2025_006
#PBS -l nodes=12:ppn=24 -l mem=235gb
#PBS -l walltime=72:00:00
#PBS -A 2025_006
##PBS -n abe
##PBS -M haopeng.wang1@kuleuven.be
#PBS -m n


cd /dodrio/scratch/projects/2025_006/COCONUT_TE/COCONUT_JobSubmission/
##module swap cluster/dodrio/cpu_rome_512
module load CMake/3.26.3-GCCcore-12.3.0
module load PETSc/3.20.3-foss-2023a
module load vsc-mympirun


mympirun --universe 288 ./coolfluid-solver --scase /dodrio/scratch/projects/2025_006/COCONUT_TE/COCONUT_JobSubmission/map_unsteadyeclipse_fullMHD.CFcase
```

Attention!

It is possible that the model would break due to memory loss or something like that. If this situation occurs, make a restart following the instruction in the time-evolving COCONUT Manu. A snapshot of this operation is illustrated below. But you need to submit the job to run “result_drawing_COCONUT_Seq.f90” shared in

··· › TestCaseAround2024MayEvent(TheLatesVersion) › Scratch_DiskSection › COCONUT_Map › results-map-res › 30Rs_lv6 › Timeevolving › fullMHD › **2024**

via:

make

qsub ./run_jobsub_Tier1.pbs

The purpose is to create the .CFmesh file, which is used for restarting, from available .plt file.

Then you can submit the TimeevolvingfullMHD.slurm. During running, it is possible that the model would break due to memory loss or something like that. In that case, you would need to change filename in the CFcase file from ‘corona.CFmesh’ to the result of last CFmesh file before break. Additionally, you also need to check the time from the **slurm-xxxxx.outfile** before break and change the line of initial time like this:

Simulator.SubSystem.InitialTime = 0.0

In code hour unite, the restart physical time *3600.0/1447.2, for first time running it should be set to 0.0

If the corresponding PhysicalTime is 1169.5009, it should be replaced as 1169.5009*3600.0/1447.2.

Also, ensure that “Simulator.SubSystem.MaxTime.maxTime”is set to at least $t_n \times \frac{3600.0}{1447.2}$. The code will terminate at this time.

Alternatively, such breakdown can be avoid by using the big memory nodes via.

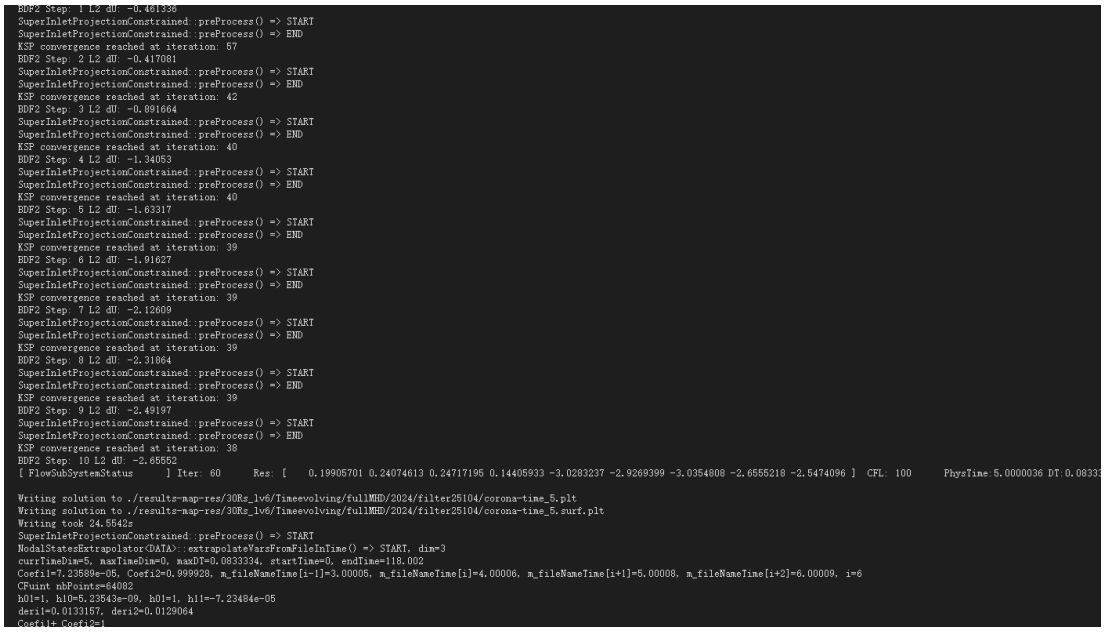
#SBATCH –partition=bigmem

#SBATCH –mem-per-cpu=20000M (We found the execution time of the big memory node should be limited to within 12 hours. We suggest to use 4 nodes, 288 CPU cores for the “30Rs_lvl5.CFmesh”)

Besides, you can reduce the number of CPU cores used per node and then allocate more memory per core by adding to your job script: ‘#SBATCH –mem-per-cpu=15000M"

The Fortran script “result_drawing_COCONUT_Seq.f90" can be used to creat a “.CFmesh" file from the corresponding “.plt" file. Keep in mind you should set “Time_sequence= .false." and “write_CFmesh = .true.” Meanwhile, you should set “Seq_Start" to the integer hour of the time you want to restart from, then you run “make" to creak the “sperical.YY" file and run the code via “sbatch jobsub.slurm".

Some log messages from the time-evolving simulation:



To run a case differently from this test, please refer to the **2.2. Complie** section to recompile, described in "DetailManuforTimeevolvingCOCONUT.pdf", and then run steps 1-3. Below is a snapshot of the content.

2.2. Complie

You can find a file named **NodalStatesExtrapolator.ci** in the path of .../the path you downloaded COCONUT/src/Framework. In the file, you need to change a few lines concerning to 'case dependent 1-5'.

1.Case-dependent_1 is determined by the structure of the file names of your magnetograms. For example, for a series of data named map_adapt_lmax15_20070909020000.dat. case-dependent_1 will be like this:

std::string FileName_prefix="./MapData/map_adapt_lmax15_2007";

std::string FileName_suffix="0000.dat";

2. In case-dependent_2, you need to specify these two following parameters. "CFuint m_magfiles" is the number of magnetograms you provided, corresponding to $t_{-1}, t_0, \ldots t_n, t_{n+1}$, where the simulation covers the time interval between t_0 and t_n. And "CFuint Cadence" is the time interval between two adjacent magnetograms (in hours).

3.In case-dependent_3, you can change the year of this simulation and days of February of this year. "CFuint year=2007" and "CFuint days_Feb=29"

4.For case-dependent_4, "CFuint data_start" is the time for starting simulation t_0, and the stucture will be like this:

CFuint data_start=2007091014 for September 10 at 14 O'clock

CFuint data_start=2007101014 for October 10 at 14 O'clock

5.In case-dependent_5, you change "current_data" to the time t_{-1}, which is "Cadence" hours previous than t_0, and the structure is the same as case-dependent_4.

Then you can get access to the path /.../COOLFluiD_Genius_New/OPENMPI. Then you run: sh compilescript.sh

Attention! There are some parameters that should be defined according to your case:

For the file "StdUpdateSolPP.cxx" located in the "\plugins\NewtonMethod" folder, you should define the boundary pressure and plasma density using the values specified in your .CFcase file. This file can improve numerical stability of both quasi-steady and time-evolving coronal simulations.