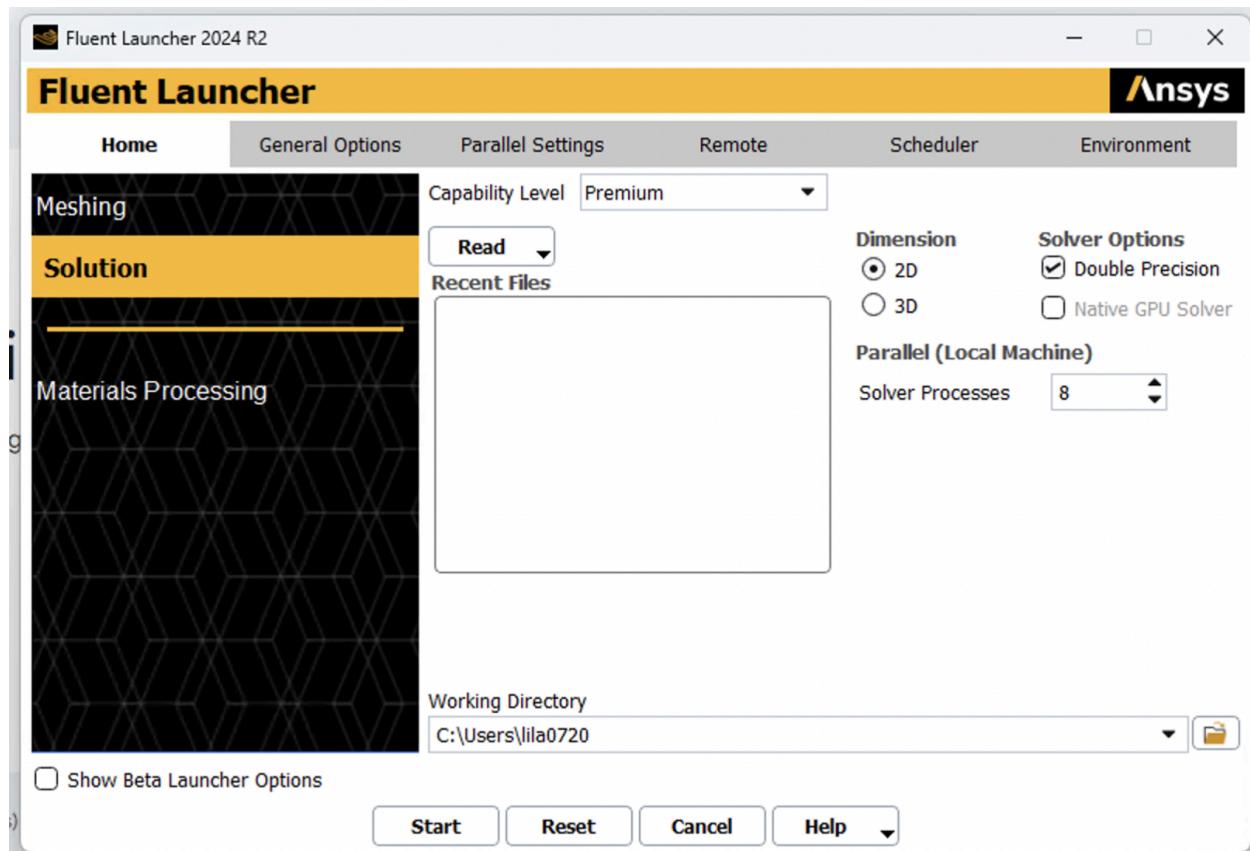


# ANSYS Fluent .POST Export Work Instruction

September 20, 2024

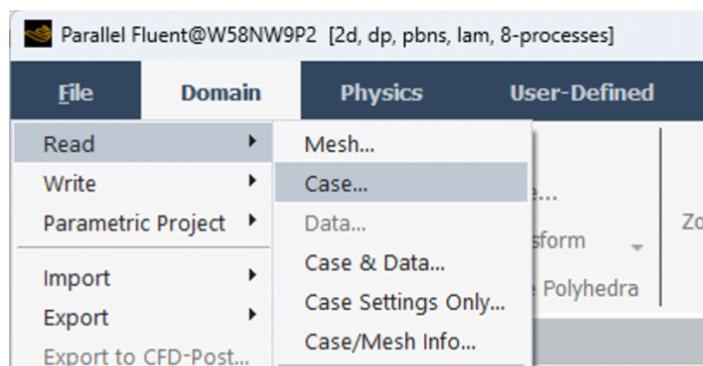
This work instruction shows how to use the script to set boundary conditions, then manually save .POST files - *there is no method yet to do everything through the script.*

1. Open ANSYS Fluent
2. Select Dimension > 2D, Solver Options > Double Precision, Solver Processes > 8. Set working directory and press Start.



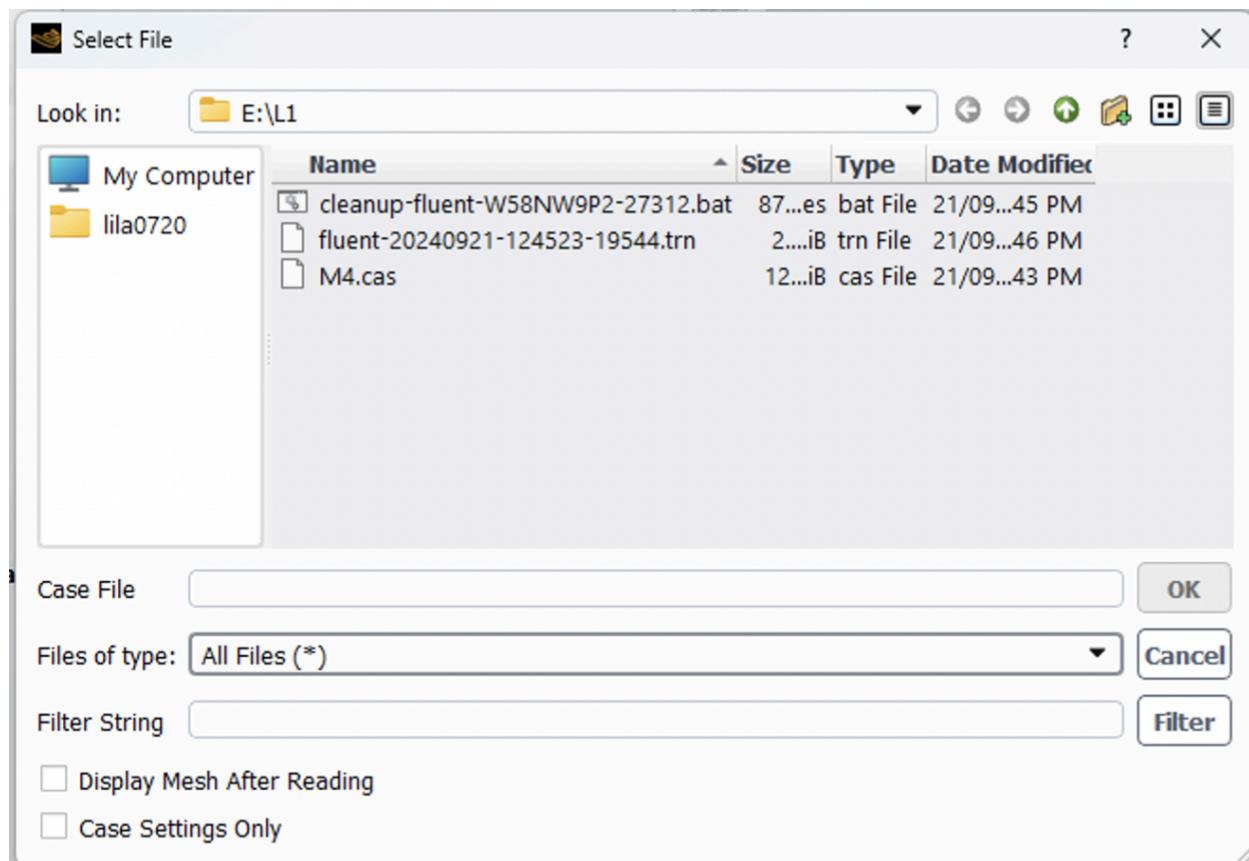
3. Close any pop-up windows that show up.

4. Go to File > Read > Case

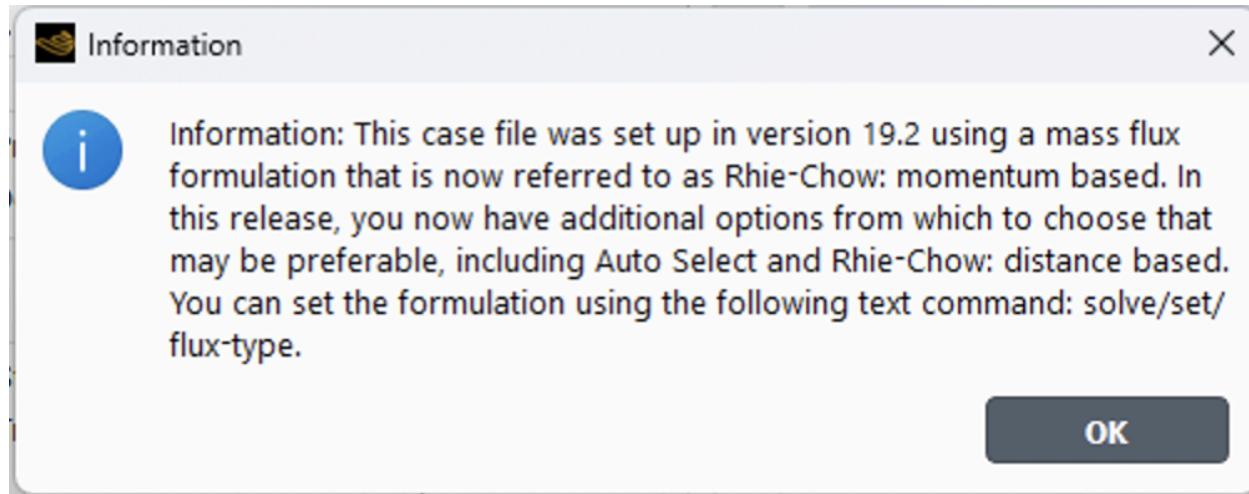


## ANSYS Fluent .POST Export Work Instruction

5. Open '**M4.cas**', the original M4 casefile. You may need to change the file type to "All Files" to see the case file:



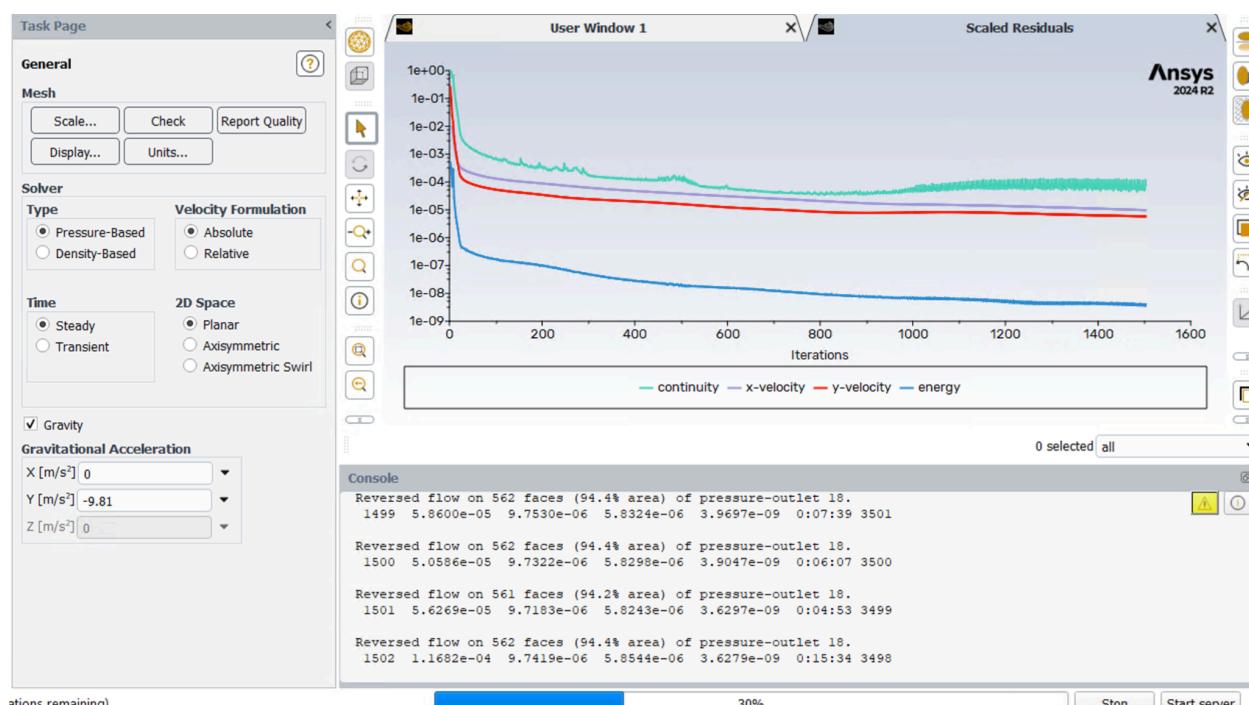
6. Press "OK" if the warning message below pops up:



7. Press the 'Enter' key in the terminal. Paste the following commands into the terminal. These set additional criteria for the boundaries on top of what's in the casefile, and solves 5000 iterations in the steady state.

## ANSYS Fluent .POST Export Work Instruction

```
define/operating-conditions/gravity y 0 -9.81
define/models/viscous/laminar y
define/materials/change-create/air/air y boussinesq 1.225 n n n n y 0.00343 n
define/boundary-conditions/modify-zones/zone-type/left symmetry
define/boundary-conditions/modify-zones/zone-type/right symmetry
define/operating-conditions/operating-temperature 298
define/operating-conditions/operating-density y 1.225
define/boundary-conditions/wall mid-bottom 0 n 0 n n n 308 n n n 1
define/boundary-conditions/wall cylinder 0 n 0 n n n 308 n n n 1
define/boundary-conditions/modify-zones/zone-type/top pressure-outlet
define/boundary-conditions/modify-zones/zone-type/left-bottom pressure-outlet
define/boundary-conditions/pressure-outlet/top y n 0 n 298 n y y n n
define/boundary-conditions/pressure-outlet/left-bottom y n 0 n 298 n y y n n
define/boundary-conditions/pressure-outlet/right-bottom y n 0 n 298 n y y n n
report/reference-values/area 1
report/reference-values/density 1.225
report/reference-values/length 1
report/reference-values/velocity 1
report/reference-values/viscosity 1.7894e-5
report/reference-values/temperature 298
solve.monitors/residual/convergence-criteria 0.00001 0.00001 0.00001 0.000001
solve/initialize/set-defaults/temperature 298
solve/initialize/set-defaults/pressure 0
solve/initialize/set-defaults/x-velocity 0
solve/initialize/set-defaults/y-velocity 0
solve/initialize/initialize-flow
solve/iterate 5000
```

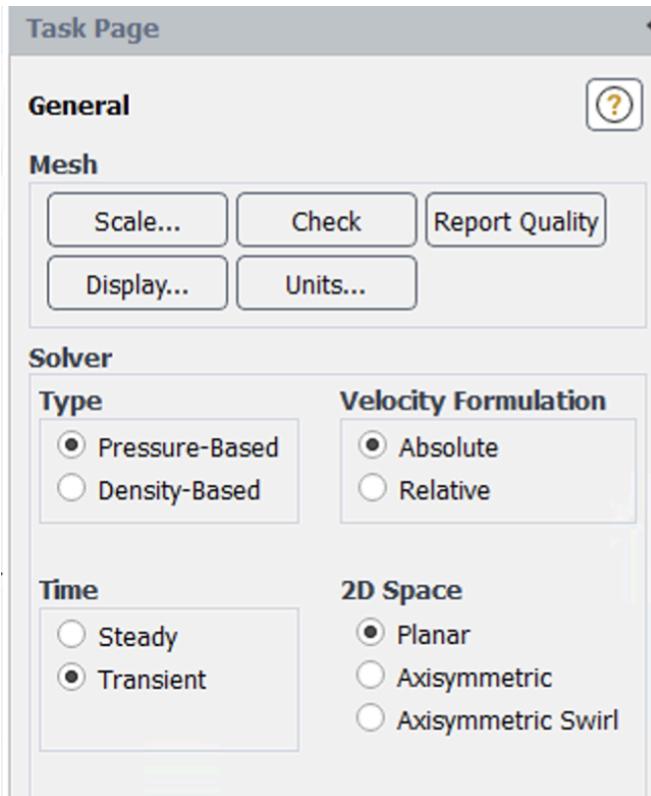


## ANSYS Fluent .POST Export Work Instruction

- Once the steady-state run finishes, paste the following commands to change to an unsteady regime, and collect the drag & lift coefficients, as well as do an internal query of 2 points in the contour:

```
define/models/unsteady-2nd-order y
solve.monitors/force/set-drag-monitor CL y cylinder () y y CL y 2 n 1 0
solve.monitors/force/set-lift-monitor CD y cylinder () y y CD y 3 n 0 1
surface/point-surface top_point 0.005 0.01
surface/point-surface bottom_point 0.1 -0.01
solve.monitors/surface/set-monitor temp-top "Vertex Average" temperature top_point () n n y
"temp-top" 1 y flow-time
solve.monitors/surface/set-monitor xvel-top "Vertex Average" x-velocity top_point () n n y
"xvel-top" 1 y flow-time
solve.monitors/surface/set-monitor yvel-top "Vertex Average" y-velocity top_point () n n y
"yvel-top" 1 y flow-time
solve.monitors/surface/set-monitor temp-bottom "Vertex Average" temperature bottom_point
() n n y "temp-bottom" 1 y flow-time
solve.monitors/surface/set-monitor xvel-bottom "Vertex Average" x-velocity bottom_point () n
n y "xvel-bottom" 1 y flow-time
solve.monitors/surface/set-monitor yvel-bottom "Vertex Average" y-velocity bottom_point () n
n y "yvel-bottom" 1 y flow-time
solve/set/extrapolate-vars y
```

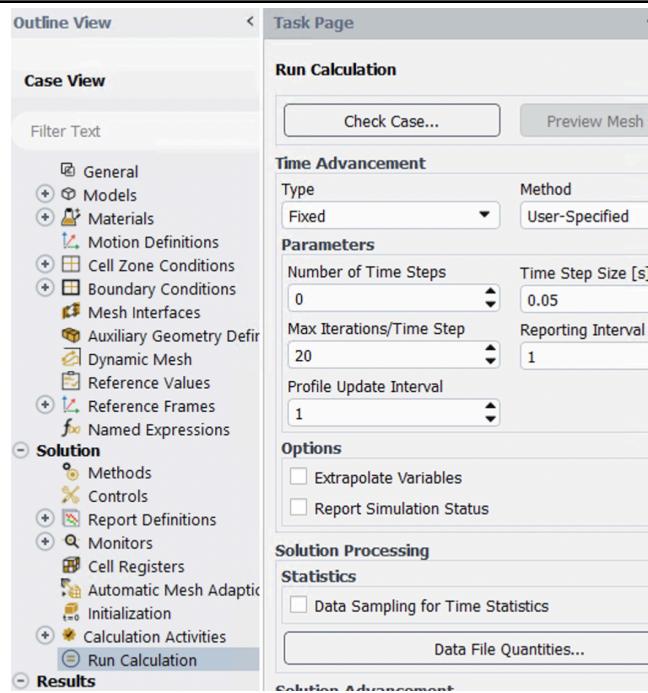
Note that this should have set the “Time” panel to “Transient”:



- Paste the following into the terminal. This sets the deltaT for the simulation as **0.05**. Check on Solution > Calculation activities > Run Calculation that the “Time Step Size [s]” has indeed been changed.

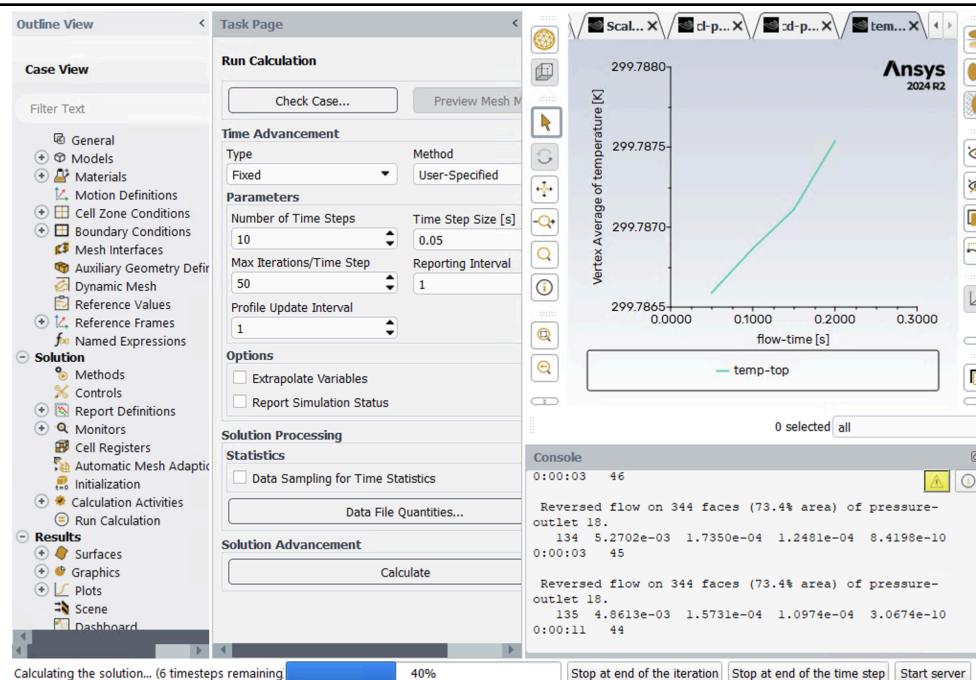
## ANSYS Fluent .POST Export Work Instruction

solve/set/time-step 0.05



10. Paste the following into the terminal. As an example, this solves 2000 timesteps at 50 max iterations per timestep. This is to run the transient calculations for a few initial timesteps to reach convergence of the cumulative mean statistics.

solve/dual-time-iterate 2000 50



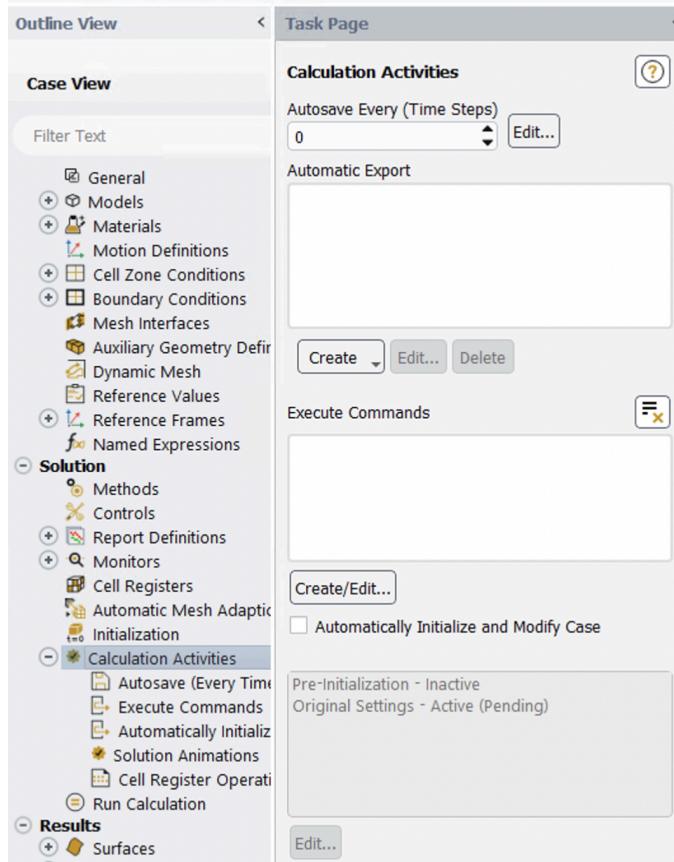
(The above image shows the command 'solve/dual-time-iterate 10 50')

## ANSYS Fluent .POST Export Work Instruction

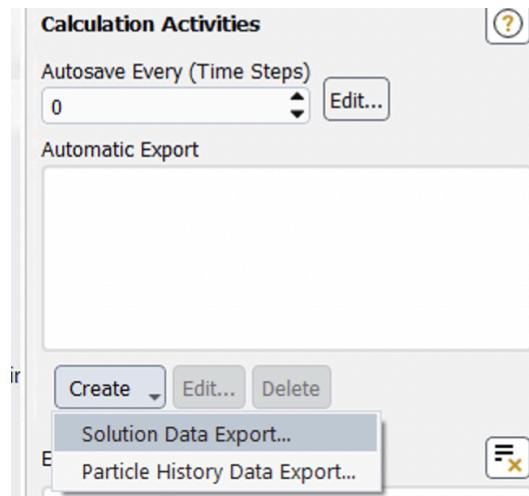
**11. IMPORTANT: Paste the following into the terminal. This ensures that NO .h5 files are saved over the duration of the simulation.**

```
file/auto-save/data-frequency 0
```

To confirm this is in place, check Calculation Activities > Autosave Every (Time Steps).

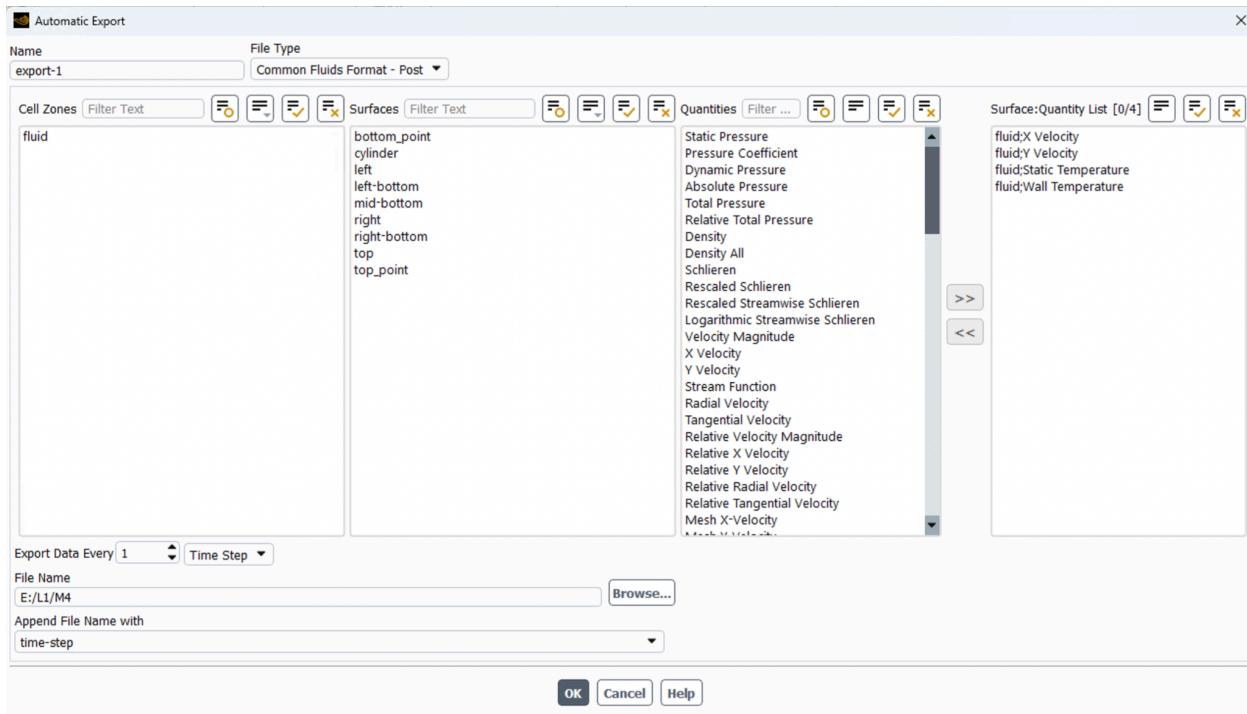


**12. Under "Automatic Export" (shown in the image above), click "Create" > "Solution Data Export"**



## ANSYS Fluent .POST Export Work Instruction

13. Click on File Type > “Common Fluids Format - Post”, then select “Fluid” under Cell Zones, and the following parameters: “X Velocity”, “Y Velocity”, “Static Temperature”, “Wall Temperature”, then the “>>” arrow:



14. Select Export Data Every “2” timesteps, then select “OK”.  
15. Paste the following into the terminal. This checks “Data Sampling for Time Statistics” and solves 20000 timesteps at 50 max iterations per timestep.

```
solve/set/data-sampling y 1 y y y y  
solve/dual-time-iterate 20000 50
```