Transient

Note: We will use Axisymmetric to reduce computational time.

1. Start Workbench 2019R2
2. Under Analysis Systems, double-click Fluid Flow (Fluent)
3. Rename it to “Diffusion (Transient)”
4. Set Analysis Type to 2D and then double-click Geometry to launch SpaceClaim
5. Sketch two concentric circles of diameter 2m and 200mm for use with the patch initialization
6. Use lines to cut circles into quadrants and delete all but the top-right quadrant
7. Exit SpaceClaim, then double-click on Mesh to launch the mesher
8. Add two Face Sizings, one to each face, Element Size = 1/10 of the radius of the face. (Soft) MAKE SURE THE UNITS ARE CORRECT.
1. Named Selections: outer_boundary, axis, symmetry, inner_region, outer_region
9. Exit the mesher and make sure Mesh is updated in Workbench window
10. Save project now and often
11. Double-click on Setup to launch Fluent
12. Change Steady to Transient, set 2D Planar to 2D Axisymmetric
13. Enable energy equation
14. Specify material properties:
    a. Thermal conductivity, density, specific heat = 1 (to give non-dimensional problem)
    b. Leave viscosity as default value
15. Domain > Mesh > Units > Temperature = C
16. Specify the boundary conditions:
    a. Outer-boundary: Wall: Heat Flux = 0
    b. NOTE: You will need to have both an axis and a symmetry boundary condition.
17. Controls > Equations > disable Flow
18. Add Volume Report
    a. Report Definitions > New > Volume Integral > Volume-weighted Average > Temperature > Static Temperature > Zones: inner_region
19. Initialization > Standard Initialization
    a. T = 0°C
    b. Patch > Apply T = 1°C to inner_region
20. Calculation Activities > Create > Solution Data Export
    a. File Type: CDAT for Ensight and CFD Post
    b. Cell Zones: both
    c. Quantities: Static Temperature
    d. Export Data Every 0.002 seconds, change Time Step to Flow Time
        i. Exporting only at the required non-dimensional times keeps the file size smaller
    e. Make sure “Write Case File Every Time” is selected
    f. Choose a File Name
        i. Note: Make sure to change the file name when you rerun the simulation with a different time-step or it will overwrite your previous results

g. Append File Name with flow-time, only need 3 decimal places

21. Run Calculation
    a. Time step: 1e-5 seconds (suggested)
        i. Make sure that this value is visibly changing with each time step, but also make sure that it doesn't reach steady state too quickly (in this case, aim for at least 100 time steps)
    b. Number of Iterations: 1000
22. Click on Calculate to have the solver calculate cell center temperatures
23. Close out of Fluent
24. Add a separate Results cell and double-click to launch CFD-Post
25. Load results and choose "Load only the last results"
26. Post-process the results in CFD Post
    a. Create a line (0,0,0) to (1,0,0) with 100 samples
    b. Plot temperature along the line
    c. Variable tab > Temperature > Units > C
    d. Import csv of analytical results
    e. Change axes titles under the respective axis tab
    f. Change series names under the Line Display tab
    g. Change font sizes under the Chart Display tab
27. Refine the mesh, re-run and check sensitivity of results to the mesh