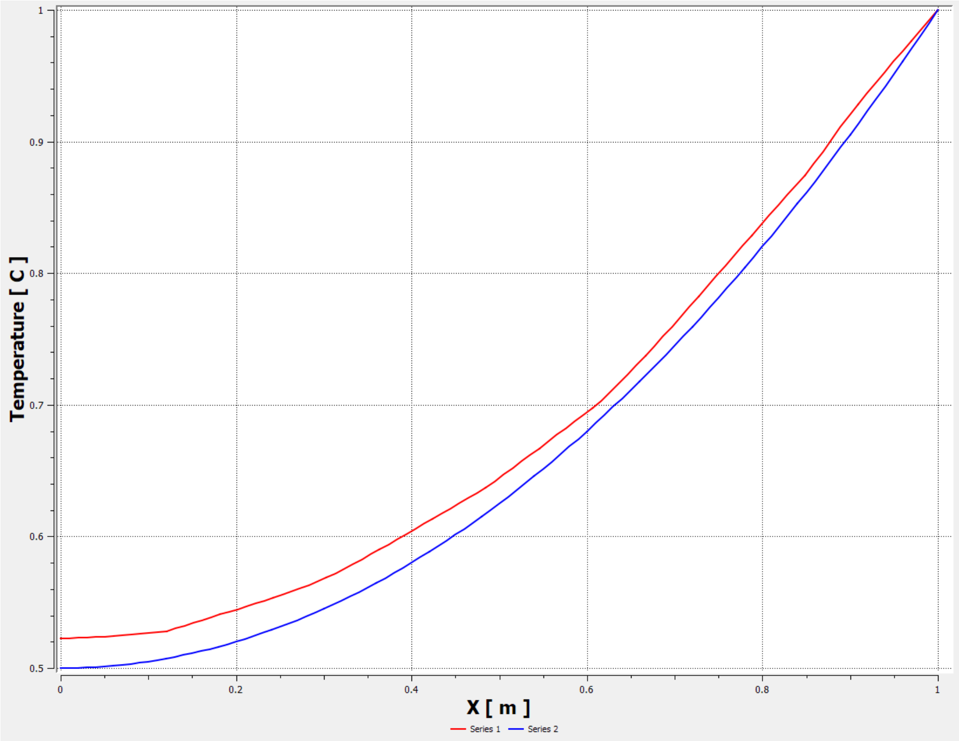
Diffusion: Outline of Steps in ANSYS

Steady State

1. Start Workbench 2019R2
2. Under Analysis Systems, double-click Fluid Flow (Fluent)
3. Rename it to "Diffusion (Steady)"
4. Double-click Geometry to launch SpaceClaim
5. Create a sphere at the origin with diameter of 2 meters
6. Exit SpaceClaim, then double-click on Mesh to launch the mesher
7. Add Body Sizing, Element Size = 0.1 meters. (Soft) MAKE SURE THE UNITS ARE CORRECT.
8. Named Selections: farfield, fluid_domain
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. Enable energy equation
13. Specify material properties:
    a Thermal conductivity, density, specific heat = 1 (to give non-dimensional problem)
    b Leave viscosity as default value
14. Specify the Cell Zone conditions
    a Enable Source Terms
    b Source Terms tab > Energy > Edit
        i. Number of Energy sources = 1
        ii. Down arrow next to gray box > Select Constant
        iii. Value = -3 W/m3 (for alpha = ½)
15. Domain > Mesh > Units > Temperature = C
16. Specify the boundary conditions:
    a Farfield: Wall: Temperature = 1°C
    b NOTE: For Axisymmetric case, you will need both an axis and a symmetry boundary condition
17. Controls > Equations > disable Flow
18. Add Surface Report
    a Report Definitions > New > Surface Integral > Area-weighted Average > Wall Fluxes > Total Surface Heat Flux > Surfaces: farfield
19. Monitors > Residual > Convergence Conditions > Add
    a Choose the report definition we just made
    b Use Iterations = 5
    c All Conditions are Met
20. Initialization > Standard Initialization
    a T = 1°C
21. Run Calculation > Number of Iterations: 20
22. Click on Calculate to have the solver calculate cell center temperatures
23. Close out of Fluent and then double-click on Results to launch CFD-Post

24. Post-process the results in CFD Post
    a Create a line (0,0,0) to (1,0,0)
    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



25. Refine the mesh, re-run and check sensitivity of results to the mesh