CIRCLE OF WILLIS Simulation Notes

Physical Model:

* Simplified model of “Circle of Willis” in the brain.

Problem Description:

* Boundary Conditions:
  + 4Velocity Inlets
    - Inlet\_VA\_left
    - Inlet\_VA\_right
    - Inlet\_ICA\_left
    - Inlet\_ICA\_right
  + 6 Pressure Outlets
    - Outlet\_PCA\_left
    - Outlet\_PCA\_right
    - Outlet\_ACA\_left
    - Outlet\_ACA\_right
    - Outlet\_MCA\_left
    - Outlet\_MCA\_right
* Blood is modelled as a Non-Newtonian fluid using the Carreau-Yasuda Model:

Where,

: efficient viscosity

: material coefficients

The representative values of human blood:

* Steady State simulation is performed using Ansys Fluent with the pressure based coupled solver.

Set Up

* Quality Check >> Mesh orthogonal quality is greater than the recommended min threshold of 0.1 confirming the mesh is of good quality.
* Physic Set Up:
  + General >> Pressure-Based, Absolute Velocity Formulation, Steady Time
  + Operating Conditions >> Normal Conditions
  + Viscous Model >> Model >> Laminar (Since the Reynold Number for this problem is low)
  + Materials/ Create/Edit:
    - Name: blood
    - Density: constant, 1060
    - Viscosity: Carreau Model:
      * Time constant = 3.313 s
      * Power-law index = 0.3568
      * Zero Shear Viscosity = 0.056 kg/ms
      * Infinite Shear Viscosity = 0.0035 kg/ms
  + Zones >> Cell Zones >> Fluid >> Change/Edit:
    - Make sure the name is changed to blood
  + Zones >> Boundaries:
    - Arterial\_wall >> Stationary wall
    - Inlet\_Ica\_left >> velocity-inlet >> momentum >> velocity mag = 0.5 (do the same for all the inlets, VAinlet, velocity mag = 0.15)
    - Outlet\_aca\_left >> pressure-outlet >> momentum >> gauge pressure = 13332 m/s
* Numerical Solution:
  + Solution: keep the current settings and make sure that [Pseudo Transient] is checked.
  + Residual monitors are set as default (for this example)
  + Definitions >> faux reports >> mass flow rate >> select all inlets and outlets
  + Initialize the solution.
  + Write 200 for the number of iterations under Run Cals column, and then click calculate
* For visualizing the flow field:
  + Contours >> New >> Arterial\_wall (Can basically have any thing you want to find in here)
* Visualizing Pathlines
  + Pathlines >> select all inlets >> 1000 steps >> 2 Path Skip
  + Select [Velocity] and [Velocity Magnitude] under [Color By]