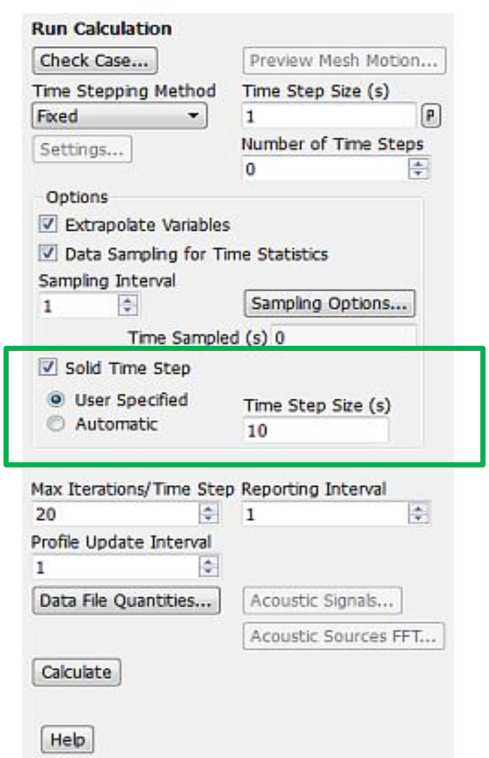
Solid Time Step

How can we reduce the time for an unsteady simulation of a Conjugate heat transfer simulation? Is there any way to give different time steps for fluid and solid in a single simulation?

Solution:

1. You can reduce the time taken for the unsteady conjugate simulation by following the steps.
2. If the flow and heat transfer are coupled (that is, your model includes temperature-dependent properties or buoyancy forces), you can first solve the flow equations before enabling energy. Once you have a converged flow-field solution, you can enable energy and solve the flow and energy equations simultaneously to complete the heat transfer simulation.
3. For transient conjugate heat transfer (CHT) problems, particularly those with combustion, the dominant time-scales in the fluid and solid zones are often quite different. In most cases, it is desirable to have a larger time step in solid zones, while maintaining a smaller time step in fluid zones. This will increase the speed at which the solid heat transfer reaches steady-state without compromising the solution accuracy of the fluid flow.
4. You can specify a solid time step on the Run Calculation task page: Solution → Run Calculation



Automatic Time Step Calculation

The default **Automatic** time step is calculated by ANSYS Fluent using the following formula:

$$\Delta t_{solid}=\frac{L_{scale\_solid}^{2}}{\alpha}$$

where,

- L_{scale_solid} is the representative length scale and is calculated as $\sqrt[3]{Solid\_Volume}$.
- α is $\frac{k}{\rho c_p}$, , the representative velocity scale, where 'k' is the conductivity, ρ is the density, and c_p is the specific heat capacity of the solid.

The calculation for Δt_{solid} results in an approximation for the solid time step which might aide in the solution running more efficiently. It should be noted, however, that this is only an engineering approximation as there is no general way to calculate a meaningful length scale for an arbitrary geometry.