Dear everyone,

I’m working on a laser-material interaction only using the “heat transfer mode”, but facing a tough problem relative to the negative temperature effect that the solved temperature is small than the initial temperature (the heat flux on one boundary is positive, the other boundary conditions are thermal insulation). I have searched in this web and found there will be three probably wrong settings, i.e. the boundary condition, mesh size/density and the time range or something esle.

In my model (heat transfer mode), there are simply only two kinds of BCs - thermal insulation and heat flux. The mesh size is “physical-controlled mesh” and “extremely fine”, and the average element quality is 0.9962. According the underlying probably wrong mistake, at first, I checked the BCs and then refined the mesh. Finally, I setting the “Study 1 --> Step 1: Time Dependent --> Study Settings --> Times” to “range (0, 1e-10, 45e-9)” , the resultant temperature is becoming smaller than the initiate temperature (293.15 K) as the solver time goes on, which is illustrated in the attachment “negative temperature 1” and “negative temperature 1 (enlarged)”. Afterward, I try to change the time step into “range (0, 1e-9, 45e-9)”, the negative temperature effect also exists. While, when the time step is “range (0, 1, 45)”, or “range (0, 1, 3)”, this problem disappears.

I’m quite puzzle about this problem. Could anyone else do me a favor to the mistake? Thanks sincerely in advance.

Ps. the “.mph” file (negative temperature.mph) is in the attachment.

Yours

FM Huang

Nov. 3rd, 2011