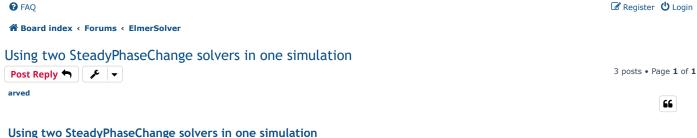
۵

"

## **Elmer Discussion Forum**



Dear Elmer community,

I am trying to solve a case involving two phase boundaries using SteadyPhaseChange. My axisymmetric setup consists of three blocks stacked on top of each other with top and bottom solid and liquid in the center. When using only one SteadyPhaseChangesolver it works well, with the case attached to this post I ran into the following error:



Is it generally possible to run such a case or would it require a modification of the solver?

I played a bit around and tried different configurations but nothing is working. It can be all found here, including a more extensive description and a couple of screenshots: https://github.com/nemocrys/elmer-double-phasechange

Thanks for your help and best regards Arved



Re: Using two SteadyPhaseChange solvers in one simulation

1 25 Mar 2022, 13:20

raback Site Admin

Looking at line ~183 in

https://github.com/ElmerCSC/elmerfem/bl ... Change.F90

You can see that there is some stuff being allocated & initialized the 1st time. These should be different for the two instances so some modifications would be needed here. Also there should be a different flag for the two different instances. You might make a copy with different name & change the flag as the most simple solution.

-Peter 0 "

## Re: Using two SteadyPhaseChange solvers in one simulation

■ 28 Mar 2022, 09:55

thanks a lot for this hint, I adjusted it and it works!

Attached to this post you can find my modified 'SteadyPhaseChange2.F90'. I'll also upload the whole setup here later today: https://github.com/nemocrys/elmer-double-phasechange

Best regards Arved **ATTACHMENTS** 

6/17/2025, 5:18 PM 1 of 2



2 of 2