How to allow heat transfer between solid and porous solid for ANSYS Fluent's thermal non-equilibrium porous model (NETM)?

Description

The non-equilibrium thermal porous model (NETM) in ANSYS Fluent can only couple a fluid zone with an adjacent solid zone and a porous zone. The model doesn't include the heat transfer between the porous solid and the adjacent zone or the blockage of the wall between the fluid and the adjacent solid for conformal meshes.

This document describes how to change the setup for Fluent versions 19.2 and later. For earlier Fluent releases, refer to solution 2053734 [1].

Solution

For a complete description of non-equilibrium heat transfer in porous media you would need to include three parts:

1. Convective heat transfer between the fluid (white zone) to the porous solid zone (blue zone, orange arrows)
2. Convective heat transfer between the fluid to the adjacent solid zone (red one, purple arrows)
3. Heat conduction between the porous solid zone (blue zone) to the adjacent solid zone (yellow arrows)

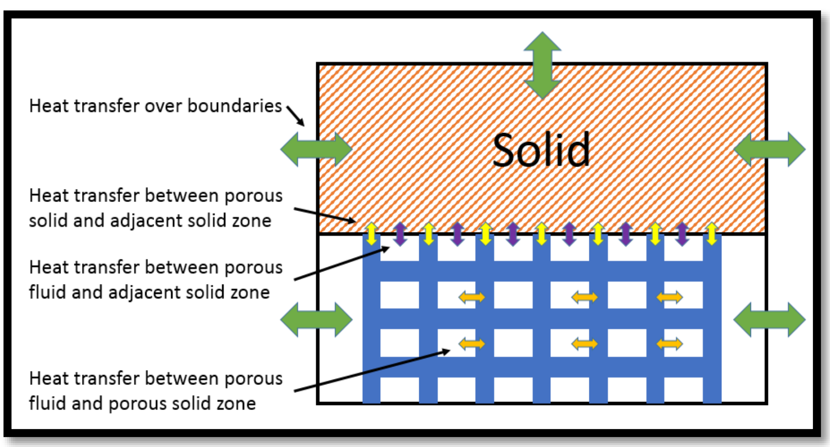


Figure 1: Zones that need to be coupled for correct description of heat transfer

By default, Fluent includes only the first two parts and neglects the heat transfer between the porous solid and the adjacent solid completely.

Typically, the thermal conduction between the porous solid and the adjacent solid is much higher than the convective heat transfer between the fluid and the adjacent solid. Still, the direct heat transfer from the fluid in the porous zone through the wall into the adjacent solid can be important, especially for a high porosity.

For a conformal mesh, it is not possible to couple all three zones (porous fluid, porous solid and adjacent solid) at a single wall. However, for non-conformal meshes you can couple them by manually creating a mapped mesh interface.

Starting from a conformal mesh

You can start from a conformal mesh, although the process is not very reliable when running Fluent on more than one CPU core. Therefore, it is recommended, that you use a non-conformal meshing approach in the first place.

For a conformal mesh, the connection between the fluid and the solid zone must be separated before creating the NETM porous solid zone. This is called slitting and is available in the ANSYS Fluent text user interface (TUI):

```
/mesh/modify-zones/slit-face-zone <name-of-one-coupled-bc-zone>
```

Important

Mesh manipulations can be tricky when using ANSYS Fluent in ANSYS Workbench. Refer to solution 2036881 [2] for more details.

Once the walls are disconnected, you can rename them and change the type of the boundary condition (BC) to interface. Then create the NETM porous solid zone and create a mapped mesh interface to couple them.

Example

The demonstration model attached to this SR is a simple tube, consisting of three fluid and one solid zone (yellow).

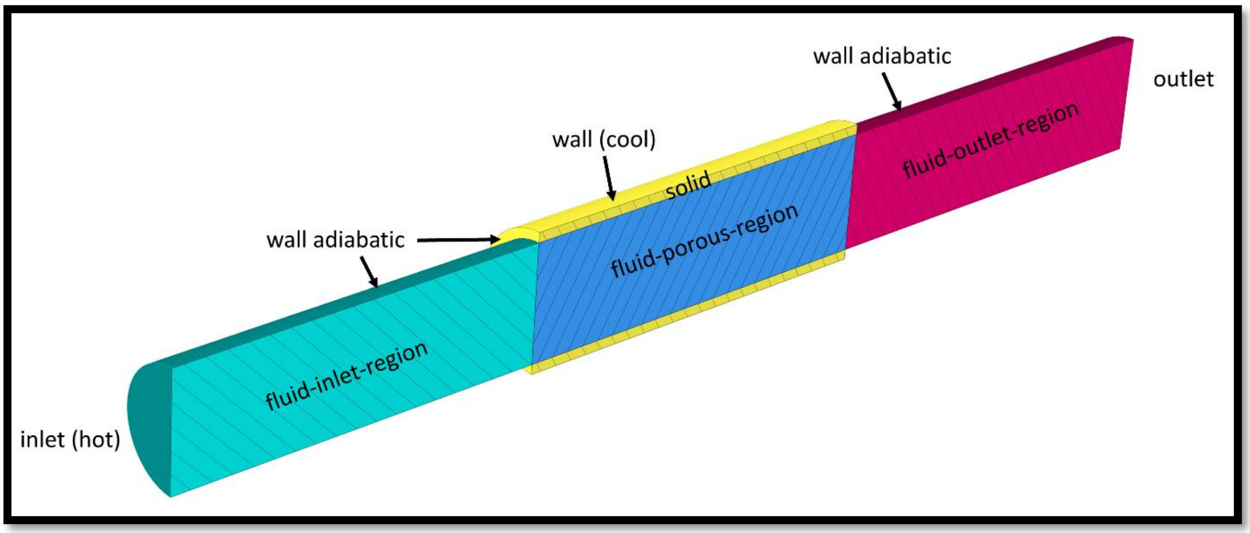


Figure 2: Demonstration model, cut view

First, check the available cell and boundary zones in Fluent.

- Cell Zone Conditions
  - fluid-inlet-region (fluid, id=8)
  - fluid-outlet-region (fluid, id=9)
  - fluid-porous-region (fluid, id=10)
  - solid (solid, id=13)
- Boundary Conditions
  - inlet (velocity-inlet, id=14)
  - interior-fluid-inlet-region (interior, id=4)
  - interior-fluid-outlet-region (interior, id=2)
  - interior-fluid-porous-region (interior, id=3)
  - interior-inlet-porous (interior, id=7)
  - interior-porous-outlet (interior, id=5)
  - interior-solid (interior, id=1)
  - outlet (pressure-outlet, id=15)
  - wall-contact (wall, id=6)
  - wall-contact-shadow (wall, id=12)
  - wall-inlet-region-adiabat (wall, id=16)
  - wall-outlet-region-adiabat (wall, id=17)
  - wall-solid-adiabat (wall, id=18)
  - wall-solid-fixed (wall, id=19)

Figure 3: Initial cell zones and boundary conditions after reading the conformal mesh

Identify the walls that need to be separated. Here, it is the only existing wall/wall-shadow pair "wall-contact" and "wall-contact-shadow".

In the Console (=TUI), type in the command to slit this pair. It is not important which of the two boundary names or ids you use. The following four lines do the same:

```
/mesh/modify-zones/slit-face-zone wall-contact
/mesh/modify-zones/slit-face-zone wall-contact-shadow
/mesh/modify-zones/slit-face-zone 6
/mesh/modify-zones/slit-face-zone 12
```

Remember that this slitting process is not very stable when running Fluent in parallel especially with distributed memory.

When the process is successful, you get two new wall zones that are named by their ID.

```
> mesh/modify-zones/slit-face-zone wall-contact

 zone 6 deleted
 zone 12 deleted
 face zone 6 created
 face zone 12 created
```

Figure 4: Fluent console input and output for slitting a conformal mesh

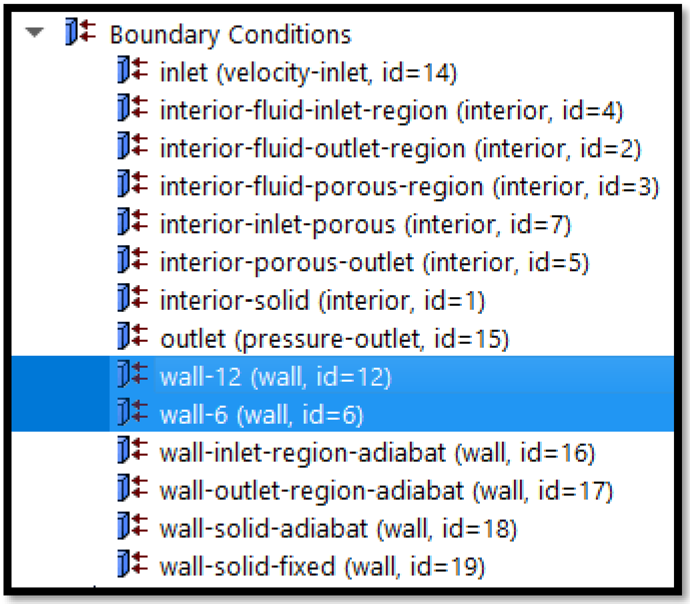


Figure 5: Generic boundary zone names after slitting

Although not required, it is recommended that you rename these zones that you can identify easily to which cell zone they belong. This can simplify post-processing considerably.

You can find out the connectivity easily by opening the BC panel for both. The cell zone they are attached to is shown in the second text field.

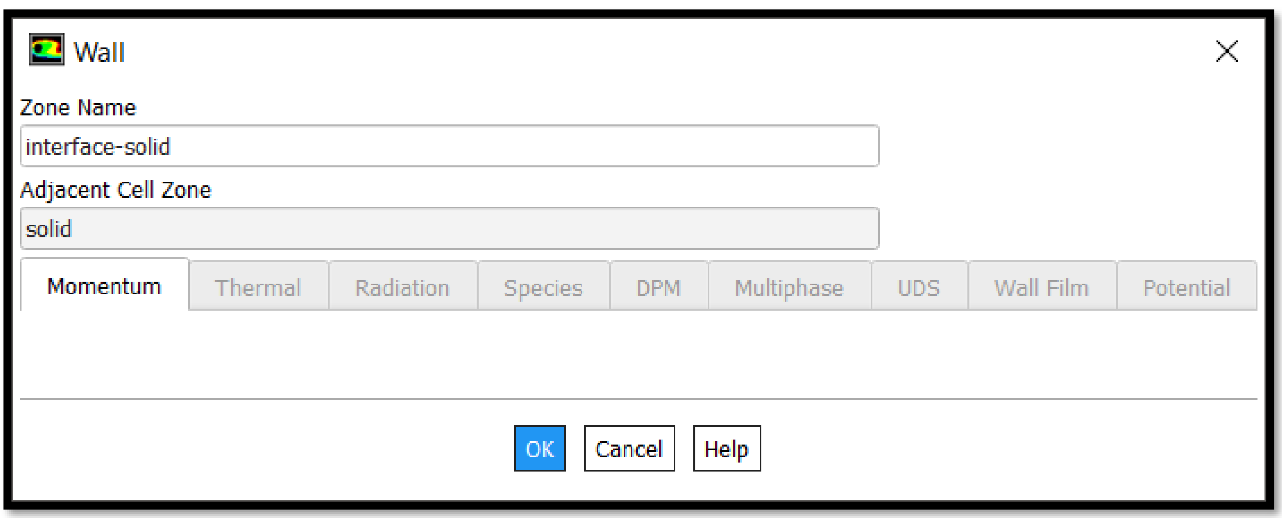


Figure 6: boundary condition panel where a zone can be renamed and to identify the adjacent cell zone

After they are renamed, change the type to interface. Note that this can also be done after NETM is activated.

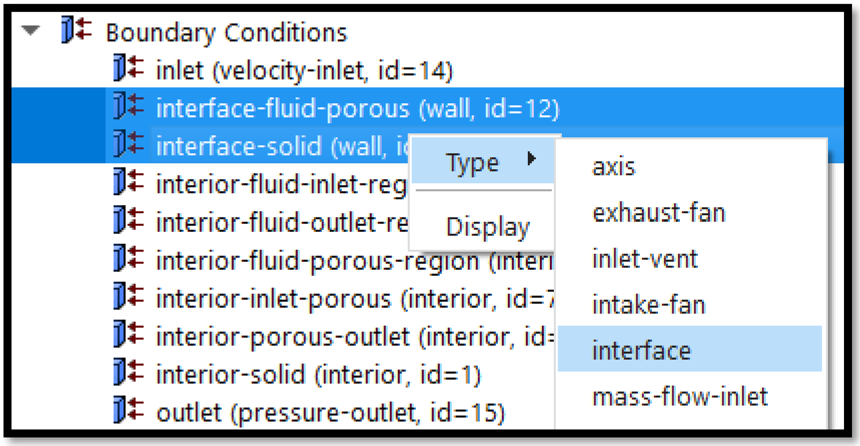


Figure 7: Changing the type of zones-to-be-coupled from wall to interface using the context menu

Next, activate the NETM porous model in the designated porous fluid zone. Of course, the model is only available if the energy equation is active. Additionally, you might need to activate the porous model, close the panel and then open it again that you can enable the model.

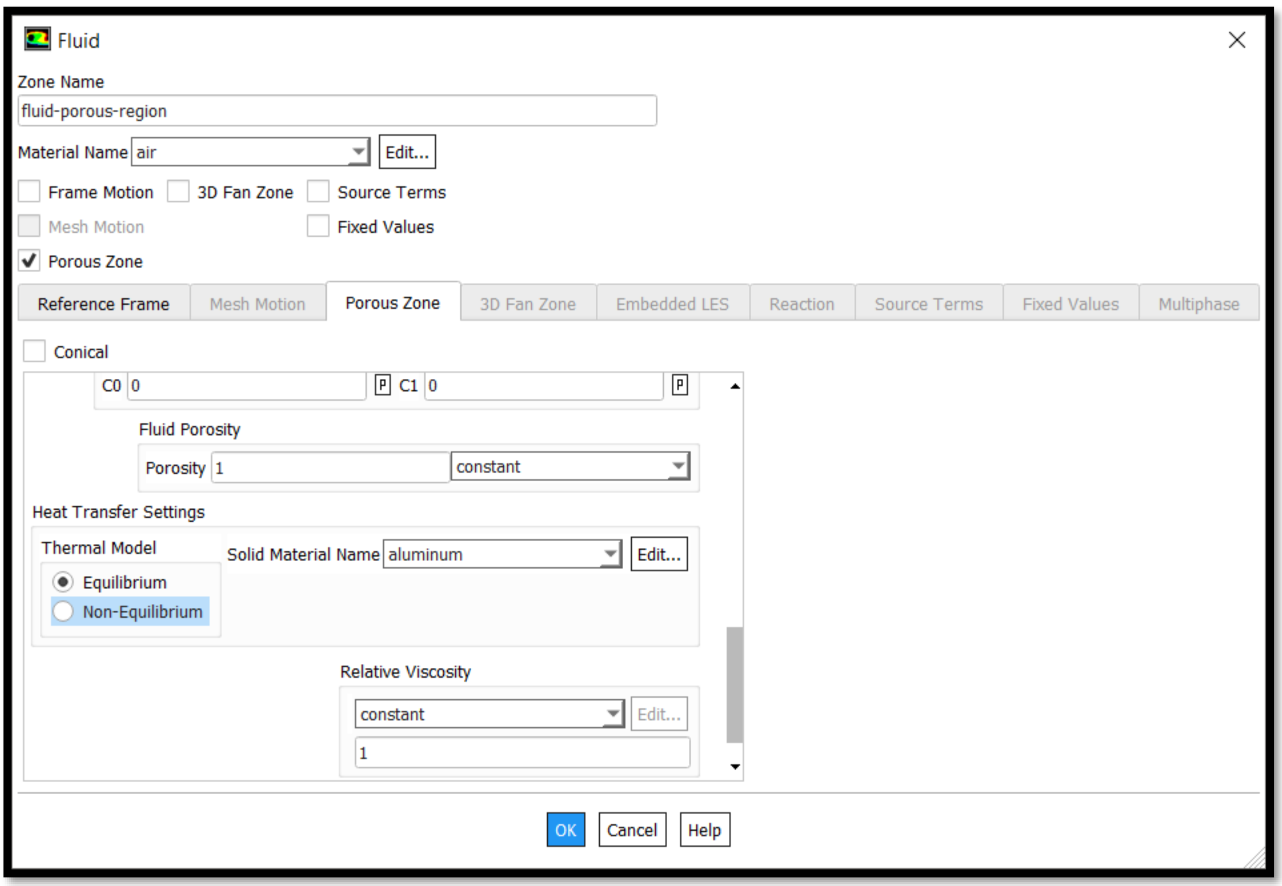


Figure 8: Activating the non-equilibrium thermal model for the porous cell zone

After you select “Non-Equilibrium”, Fluent asks it should create the required solid zone which must be answered with Yes.

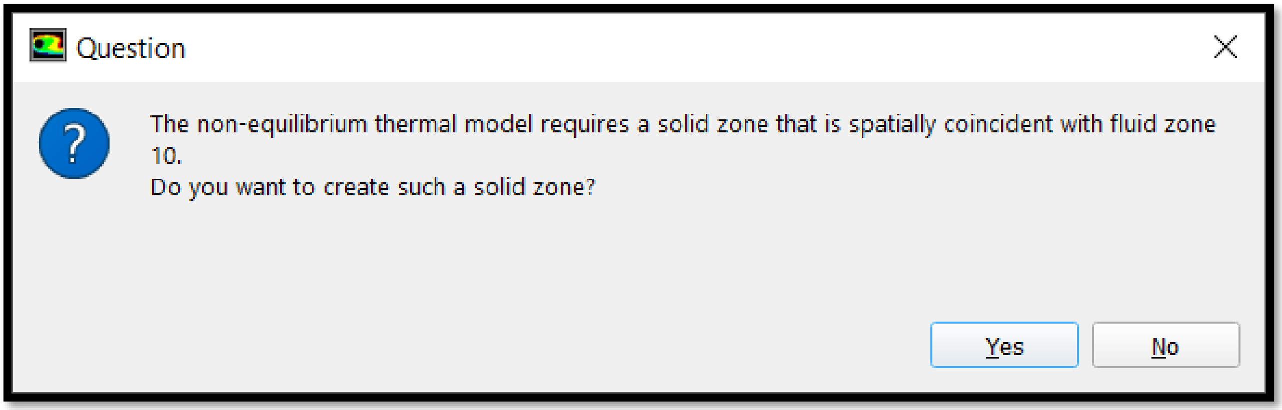


Figure 9: Create the additional cell zone for NETM

This create a new cell zone for the porous solid. It gets the same name as the fluid zone with the id as suffix. All boundaries of this solid zone are adiabatic walls by default.

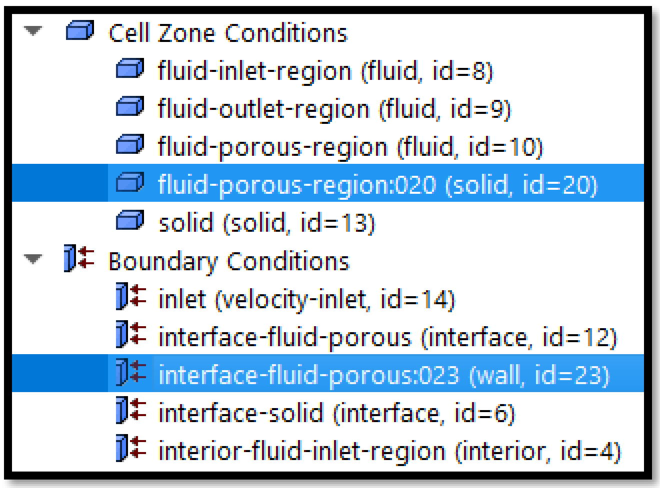


Figure 10: New zones visible in the tree after creating the new solid cell zone

Therefore, you must turn the wall zone into an interface BC again. While at it, you can also rename it that it is easier to identify the zone for post-processing.

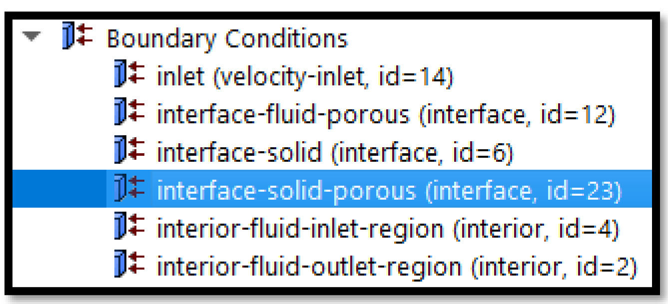


Figure 11: Three coincident interface zones required to allow heat transfer between solid, porous solid and porous fluid at the wall

Now it's time to couple the interfaces. Disregard the button Auto Create and go straight to the manual interface creation.

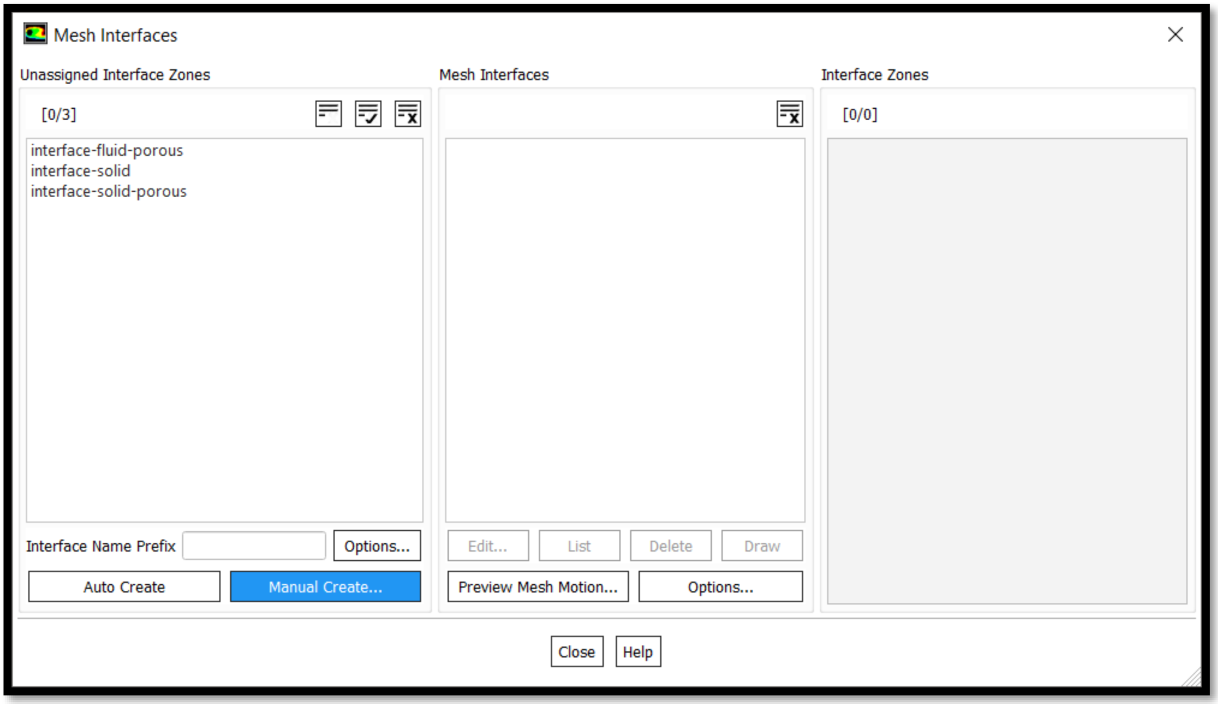


Figure 12: Mesh interface panel before creating the coupling

Specify a name, select the zones and activate the interface option mapped which enables coupled with it.

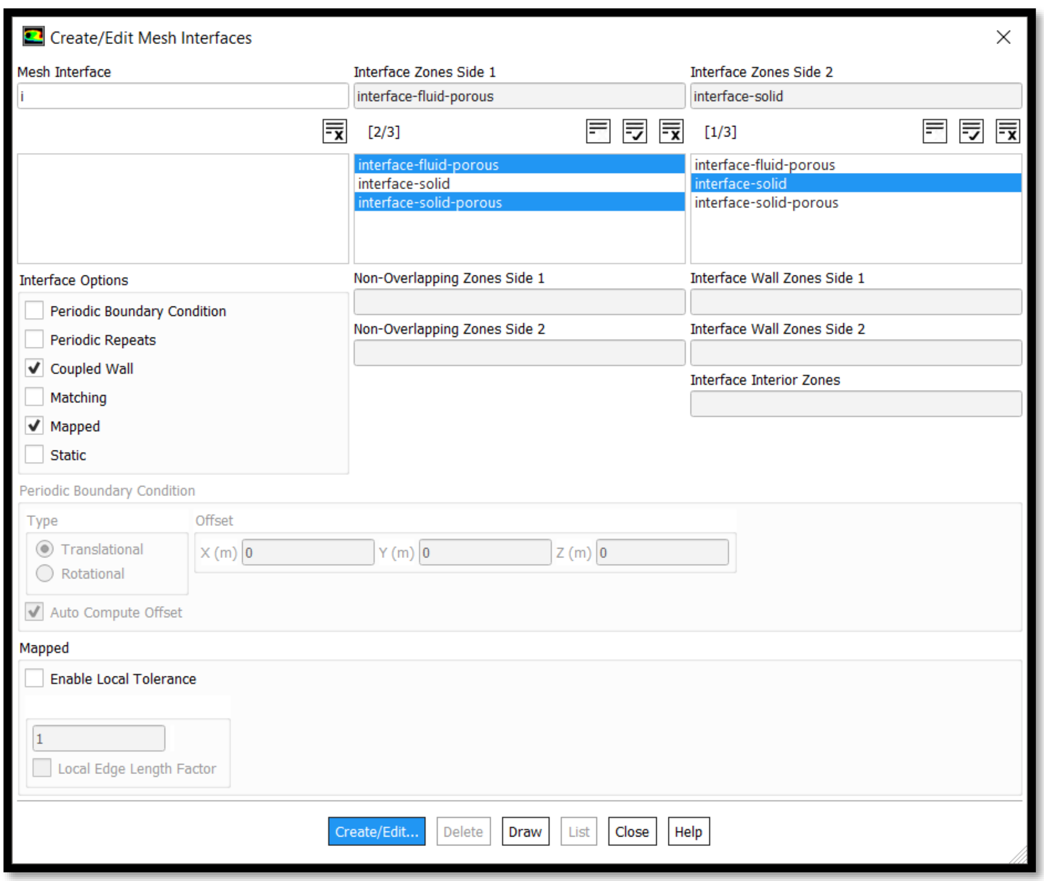


Figure 13: Panel for the manual creation of the non-conformal mesh interface with all required settings

Ignore the notification of the interface zones. The panel only shows the cell zone that is adjacent to the first interface boundary that you select.

Finally, click on Create/Edit and close the panel.

Now you can setup the rest of your simulation, run and post-process it as you like. All three zones are coupled, and the porosity is also considered at the walls which can play a very important part in accurate prediction of heat transfer.

Note: The thermal coupling of three zones can influence the convergence negatively. Ensure that the energy equation converges before contacting the ANSYS technical support. The energy residual can be a first hint, usually it must drop below 1e-8 or even lower which requires running Fluent double precision in most cases. Additionally, check the total energy balance and the energy balance across the interface. Typically, the interface itself is not a problem but the non-equilibrium approach itself can require many iterations to converge, especially for a combination of high porosity and high heat transfer between the two porous zones.

Starting from a non-conformal mesh

The process is comparable to the approach described above. Obviously, the slitting-step is no longer required which makes the process robust, at least if you are using Fluent standalone outside of ANSYS Workbench.

However, if you use ANSYS Meshing in ANSYS Workbench, you can use contact detection to automatically create non-conformal mesh interfaces in Fluent.

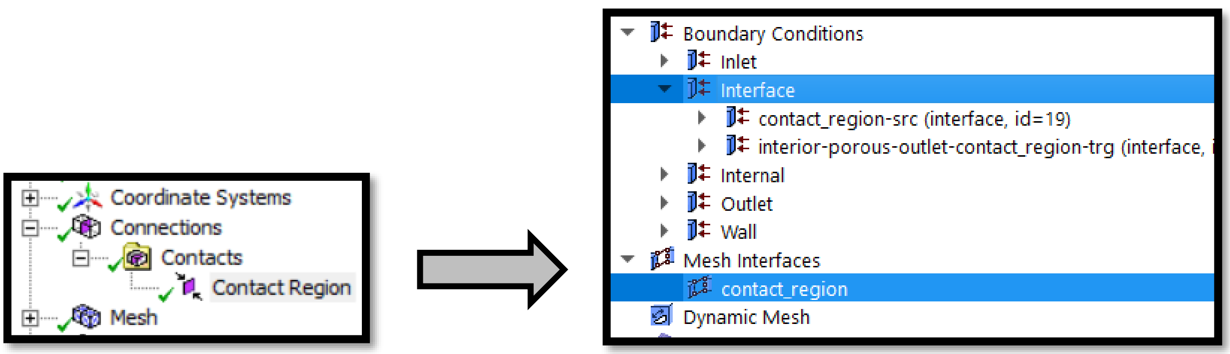


Figure 14: Contact regions in ANSYS Meshing are converted into active mesh interfaces in ANSYS Fluent automatically

Apparently, this works only for the fluid-solid coupling of two zones, since the porous solid zone doesn't exist while reading the mesh.

You must delete the interface and manually create it again to successfully couple the three interface zones, since it's not possible to edit the boundary zones of an existing mesh interface.

NETM for Fluent in Workbench

If you use the described workflow in Fluent in Workbench, you see at least one warning message that mesh modifications can't be saved for future calculations:

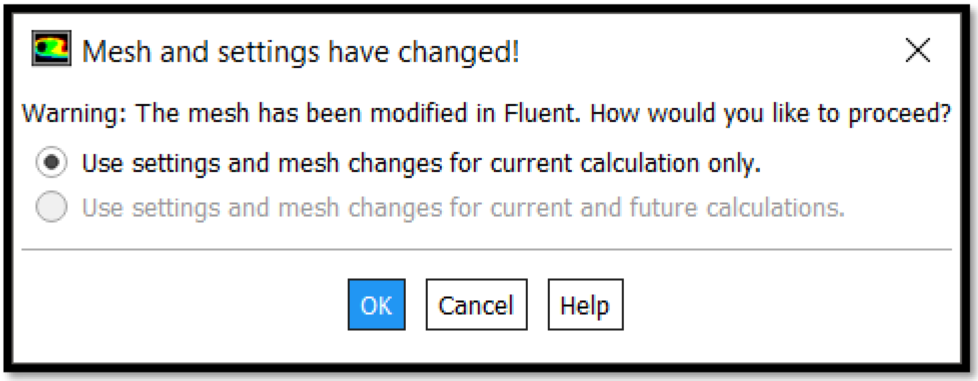
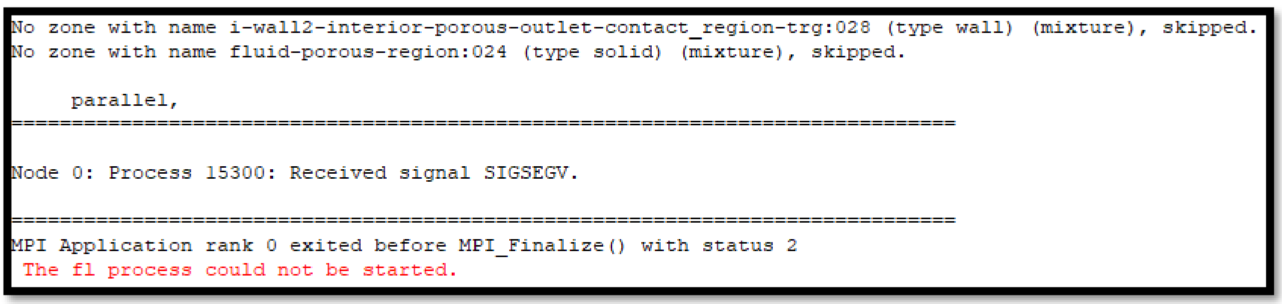


Figure 15: Warning message that mesh changes can't be used for future calculations

The first calculation should run fine, and it seems that all changes are saved to the settings file, nonetheless. But it is likely that Fluent crashes after a change to an upstream cell (geometry or mesh).



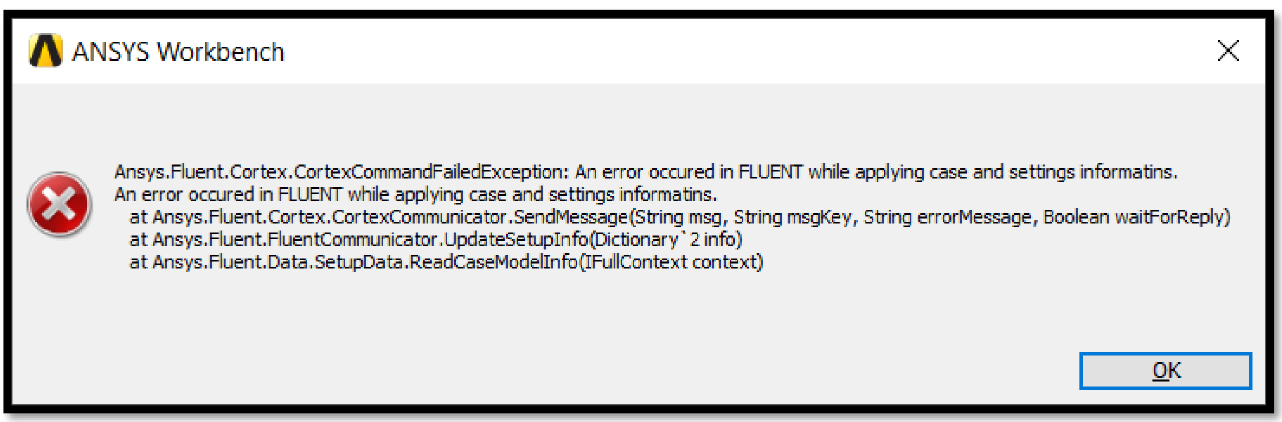


Figure 16: Termination of Fluent inside Workbench after refreshing the setup cell with a new mesh

The reason for this crash is the missing zones. With NETM, one additional solid zone is created, and the mesh interface includes one of the associated boundary zones. These zones don't exist before applying the settings which leads to a fatal error and the termination of the Fluent processes.

You can try if you can adjust the process described in solution 2036881 [2] for this workflow. But for now, the recommendation is to avoid using Fluent in Workbench when you need the coupling of all three zones.

Note: The same applies in the standalone version of Fluent. When you have an existing NETM setup that you want to apply to a different mesh, Fluent can crash. Both approaches, settings file and GUI (reading a mesh over a loaded case without discarding the settings), will fail. The process is the same as in Workbench. The only workaround is to disable NETM, delete the porous solid cell zone, write the settings and then reapply NETM again after reading the settings file for the new mesh.

General notes regarding convergence

When you use NETM with or without the coupling of the three walls, you should pay close attention to the energy convergence and the overall energy balance. It is strongly recommended, that you run Fluent only with double precision and converge the energy residual below 1e-8.

Additionally, check the energy balance of the mesh interface, which should be close to zero, and the overall energy balance. Especially when you have a case with high porosity (close to 1) and a high heat transfer between the porous fluid and porous solid regions (e.g. a very large interfacial area density), it can take many iterations for the energy balance to reach an acceptable state. Often, the equilibrium model could be used instead if these conditions are met.

When assessing convergence, never trust the residuals exclusively. Always combine them with reasonable monitor quantities (temperature, energy balance, …). For transient simulations, check these monitors not just at the end of the time step but for every iteration to spot possible convergence issues early.

Attachments

2056744.zip – Example cases

References

[1] ANSYS, Inc., "ANSYS Solution 2053734: How to allow heat transfer between solid and porous solid for ANSYS Fluent's thermal non-equilibrium porous model (NETM)?," ANSYS, Inc., 23 10 2018. [Online]. Available: https://support.ansys.com/AnsysCustomerPortal/en_us/Knowledge%20Resources/Solutions/FLUENT/2053734. [Accessed 23 10 2018].

[2] ANSYS, Inc., "ANSYS Solution 2036881: How to use the thermal non-equilibrium model for porous zones in Fluent and Workbench?," ANSYS, Inc., 21 10 2015. [Online]. Available: https://support.ansys.com/AnsysCustomerPortal/en_us/Knowledge%20Resources/Solutions/FLUENT/2036881. [Accessed 26 03 2018].

Keywords: ANSYS Fluent; NETM; non-equilibrium; thermal; porous; model; solid; coupling; CHT; conjugate heat transfer

Contributors: Akram Radwan; Dr. Amine Ben Hadj Ali