

Overset Meshes

New Capability in Fluent 18

Tom Chadwick

July 5, 2017

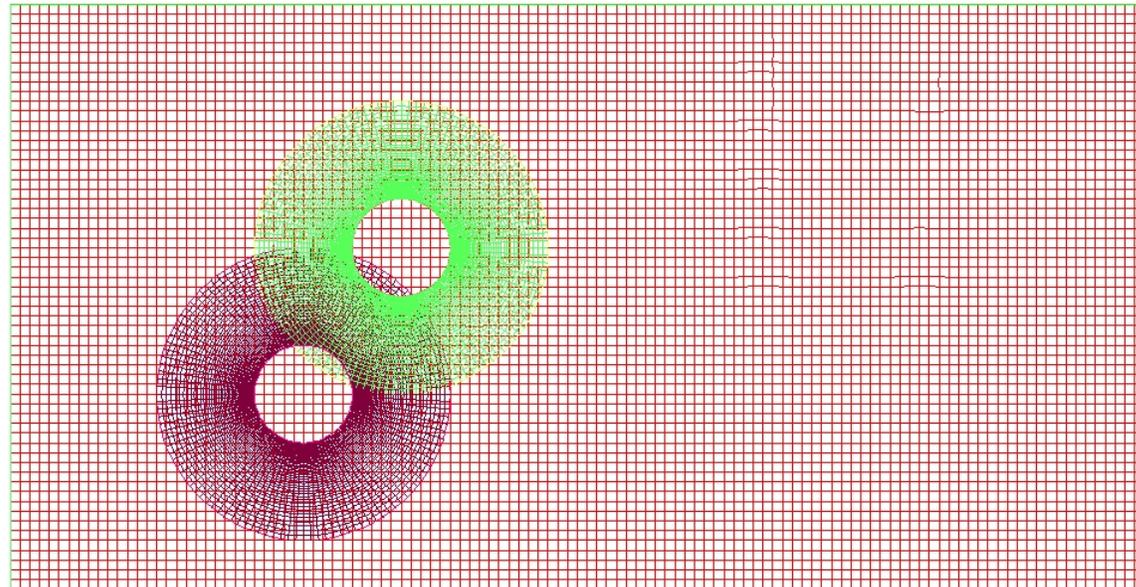


Overset Meshing

This tool allows you to build multiple meshes, one for each region of the problem, and combine them into one complete domain.

Each region is meshed in a separate Workbench project cell. All the meshes are fed into one Fluent cell.

There is one overall mesh for the whole domain which is set to be the background zone. The other meshes are defined to be component zones.

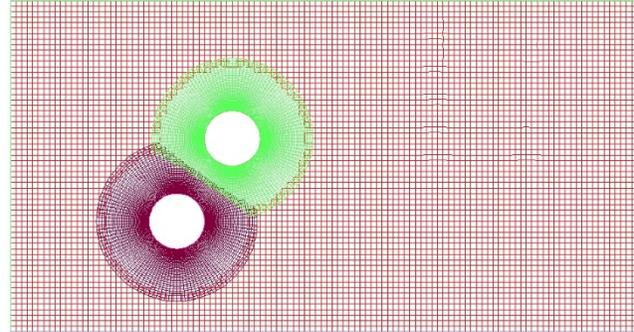


0 0.02 (m)

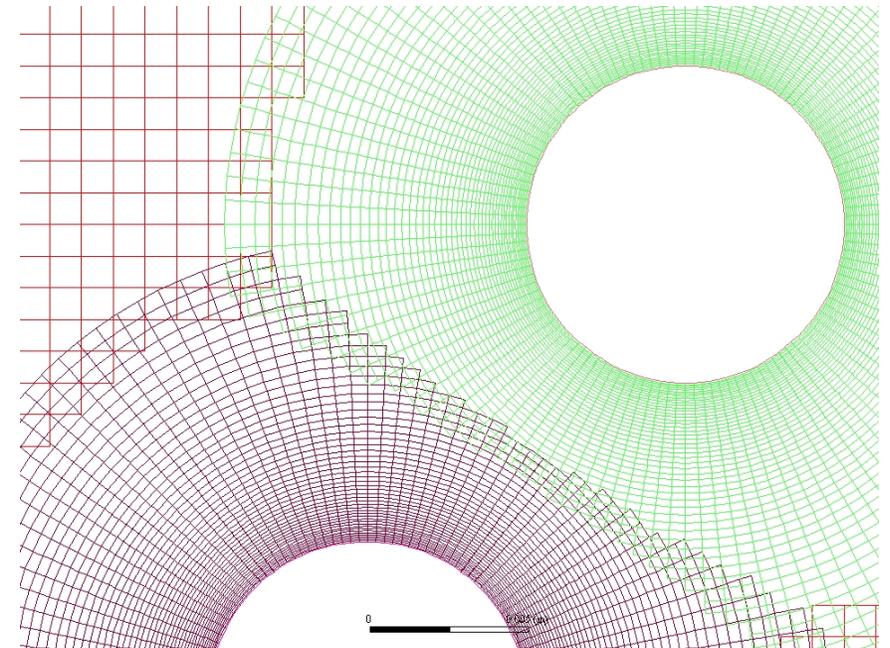
Overset Meshing

The overset mesh controls allow you to control how the meshes are combined. You can set it determine the interface location based on the relative cell sizes of the adjacent regions or based on the distance from the nearest wall boundary.

It also allows you to set which mesh domains should have priority over the adjacent ones. Assigning a higher priority to an individual zone means that this zone will be maintained over the adjacent ones.



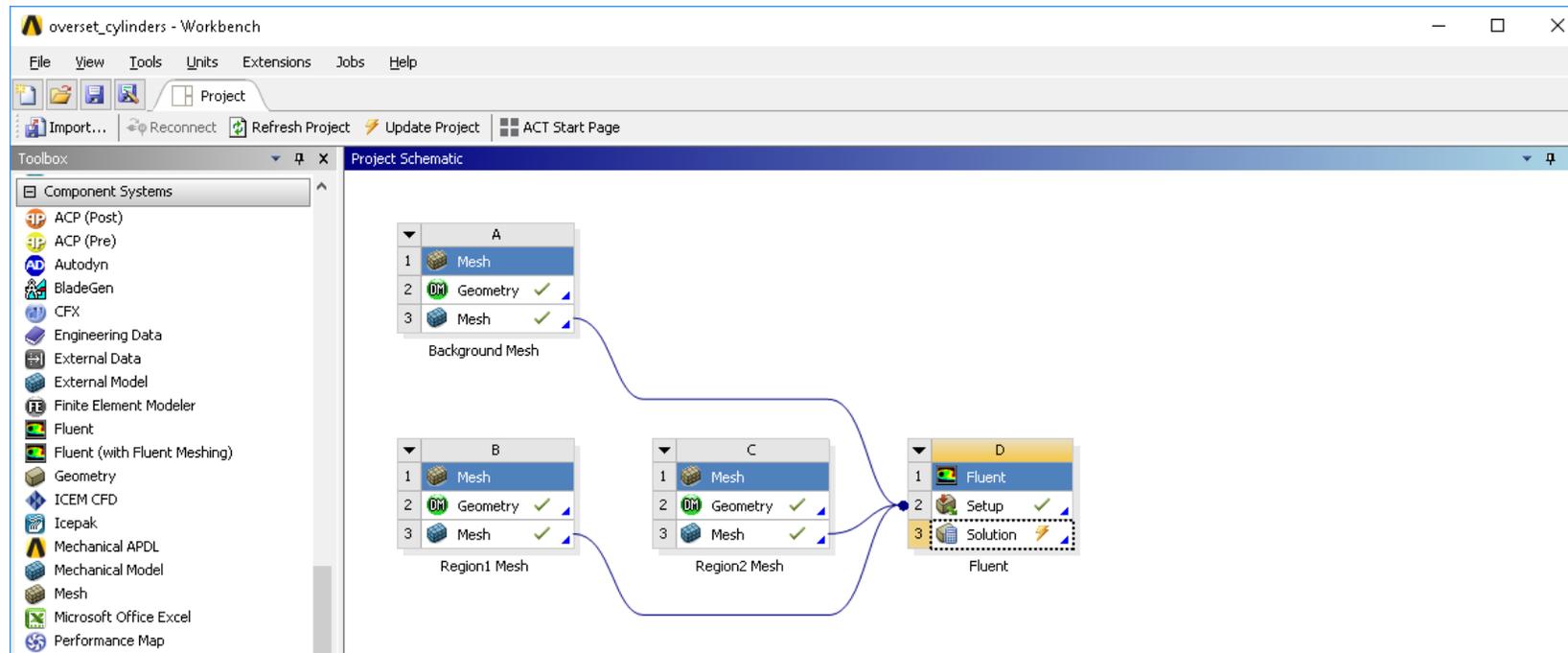
0 0.02 (m)



0 0.02 (m)

Overset Mesh Workbench Project Schematic

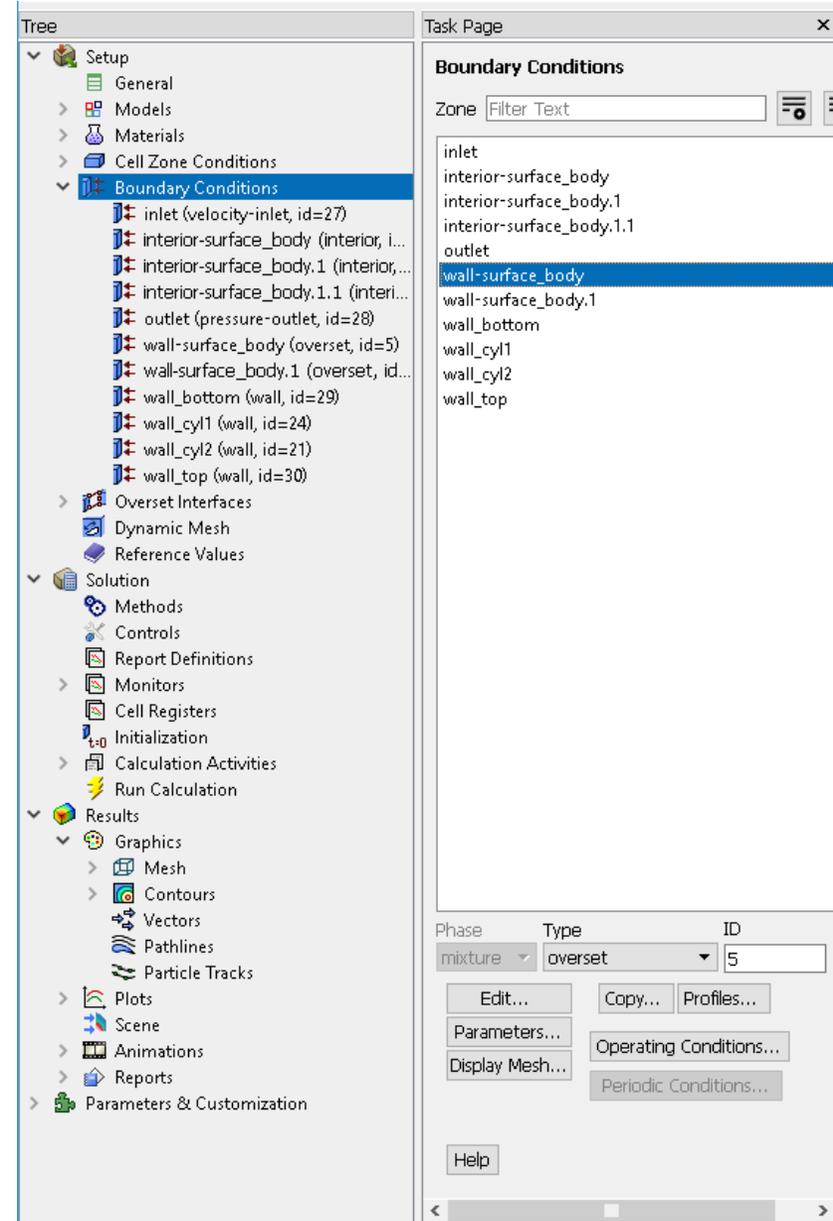
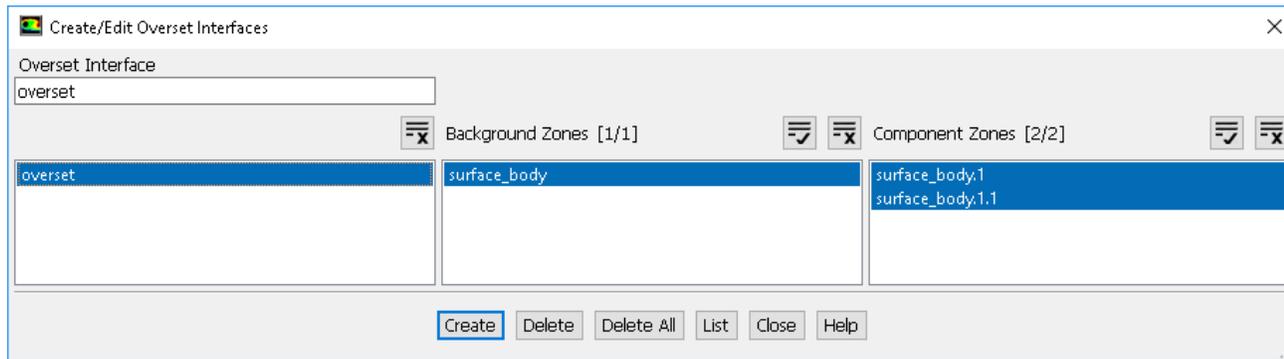
The background mesh and each component mesh is defined as a separate geometry and mesh group. All of them are connected to the Fluent setup cell.



Overset Mesh Setup

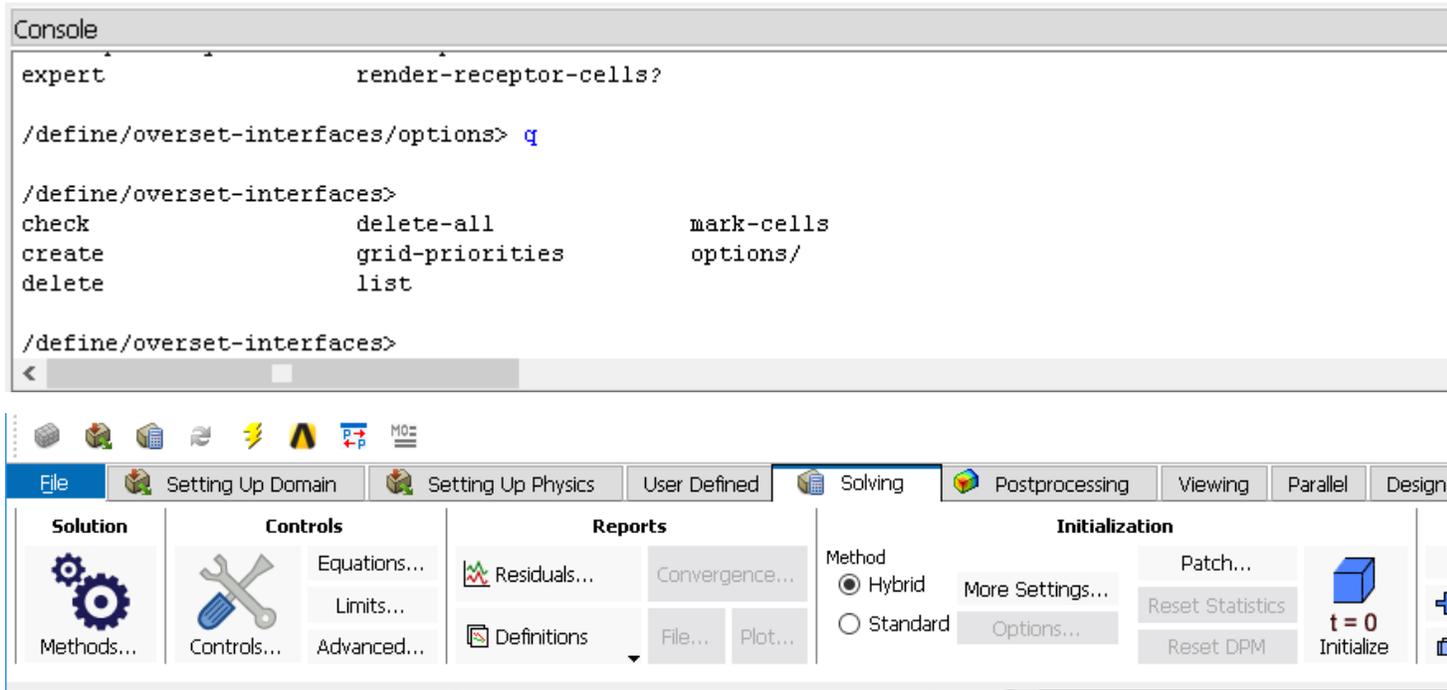
When the mesh zones are read into Fluent, the outer boundaries of the component zones are marked as walls by default. To enable the overset mesh tools, you convert them from wall to overset.

Once that is done, the tool to define the overset interface will be enabled. In this tool you set the name for the overset interface and then mark the background and component zones that will be used in the interface.



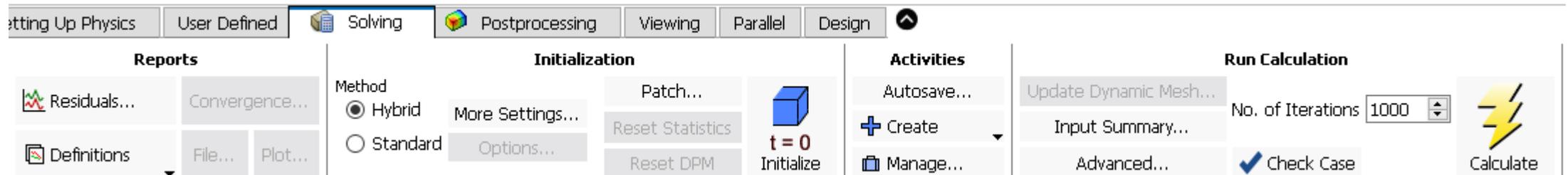
Overset Mesh Setup

The remaining controls for the overset interface are defined using text commands in the TUI. In the Fluent console go into the `/define/overset-interfaces` sub-menu. In that are the controls for the options to control how the interface is determined and set which zones have the highest priority. When the interface definition is changed, go to the Solving section and select the initialize button to update the overset interface. After re-initialization, display the mesh again to see how the interface has changed.



Solving and Post-Processing Overset Mesh Problems

Once the overset mesh has been defined and the boundary conditions have been set, the Fluent solution process is the same as for any other problem. Go to the Solving tab, set the number of iterations, then select the Calculate button. Post-processing is independent of the overset mesh.



Overset Mesh Limitations

- **The overset interface cannot contain solid cell zones.**
- **Component meshes cannot connect to a non-conformal interface.**
- **Background meshes cannot have non-conformal interfaces between them.**
- **Component zones cannot have periodic boundaries.**

Note: It is recommended to use double-precision for all overset mesh problems.