

Fluent to OpenFOAM

<https://openfoamwiki.net/index.php/Fluent3DMeshToFoam>

<https://www.cfd-online.com/Forums/openfoam-meshing-other/97471-converting-fluent-mesh-openfoam-standard-mesh.html>

<https://www.youtube.com/watch?v=f9-GDWLKixg>

This is a quick tutorial showing you how to get ANSYS Fluent mesh files (.MSH) from the ANSYS software without needing an ANSYS Fluent license. The .MSH file is then in a suitable format for use in OpenFOAM CFD Software. This gives you the opportunity to work with the true geometry from ANSYS rather than using the typical tessellated STL geometry representation that is commonly loaded in OpenFOAM.

<https://www.youtube.com/watch?v=l01awpwOcho>

<https://www.youtube.com/watch?v=dlqmFr1fZ1M>