Open Source (OS) Engineering Software

- Modeling steps with OS software
- OpenFOAM on Odyssey via Singularity container (primarily CFD/Thermal Simulations)
- Modeling in the cloud via web with open source components
  - Simscale

Modeling steps with OS software

Steps to perform engineering simulations via an open-source software are:

- Create the structure or component on which simulation needs to be done (typically using a CAD software).
  - Eg: FreeCAD (https://www.freecadweb.org/), Salome (http://www.salome-platform.org/) or Salome-Meca (https://www.code-aster.org)
- Discretize (or “Mesh”) the structure for numerical analysis
  - Eg: gmsh (http://gmsh.info), Salome (or Salome_meca)
- Prepare the case for simulation (Set up the simulation: import the geometry/mesh, specify physics, materials, initial conditions, boundary conditions etc.) and run simulation.
  - Eg: OpenFOAM, Code_Aster, Elmer
- Visualize and analyze the results.
  - Eg: Paraview (https://www.paraview.org/)

OpenFOAM on Odyssey via Singularity container (primarily CFD/Thermal Simulations)

While there are multiple OS software for CFD, we concentrate on OpenFOAM (https://en.wikipedia.org/wiki/OpenFOAM). This is a software that has been developed since the 80s and has recently seen an uptick in the number of users. Multiple commercial entities are using it as well as providing CFD simulation services around it (Eg: Simscale). For more information, visit:

https://openfoam.org/

OpenFOAM on Odyssey and Linux Desktop

OpenFOAM - Modeling Basics

The software does not come with a GUI based pre-processor. Hence at least the geometry creation needs to be done using a different software unless it is very simple. Meshing can be done within OpenFOAM, but it may be more convenient to do it outside OpenFOAM.

Modeling in the cloud via web with open source components

There are a few companies that are beginning to offer simulation services in the cloud using OS engineering software. Simscale is one such company.

Simscale

A commercial company that uses open source CFD (OpenFOAM) and Structures/Solids (primarily: Code_Aster) software.

https://www.simscale.com

For details on open source software in simscale, visit:

https://www.simscale.com/open-source/

Accounts are free (as of this writing). However, the projects will be public. Simscale has an Academic-program that allows for private academic projects:

https://www.simscale.com/academic-program/