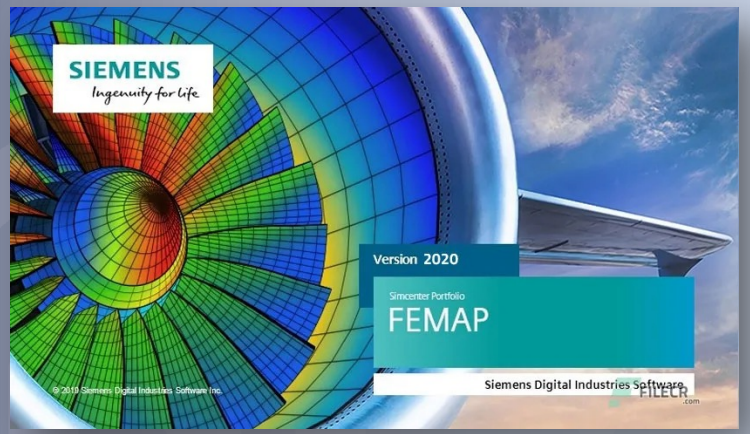
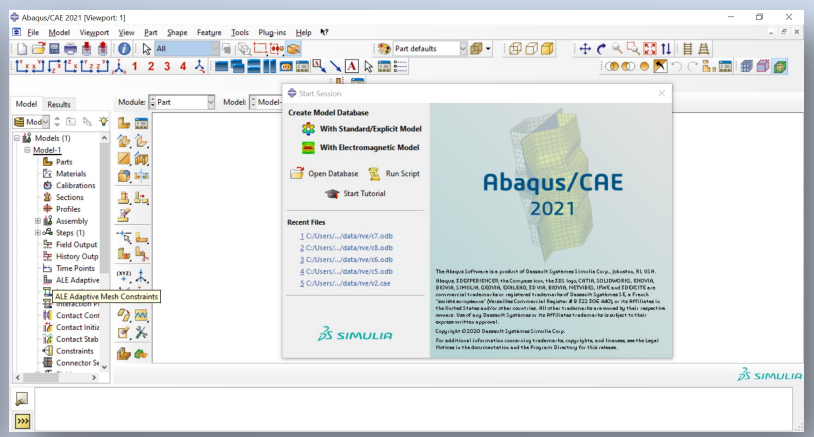
PROJECT MINERVA

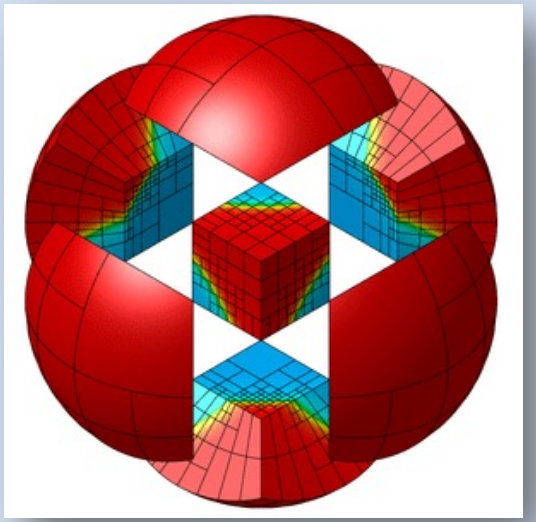
Accelerated Deployment of MFEM Based Solvers in Large Scale Industrial Problems Topic: 2/a

INTRO

- Leverage mature, commercially available CAD and FE pre/post software to develop MFEM models
- Disrupt current commercial software/hardware models for HPC FEA





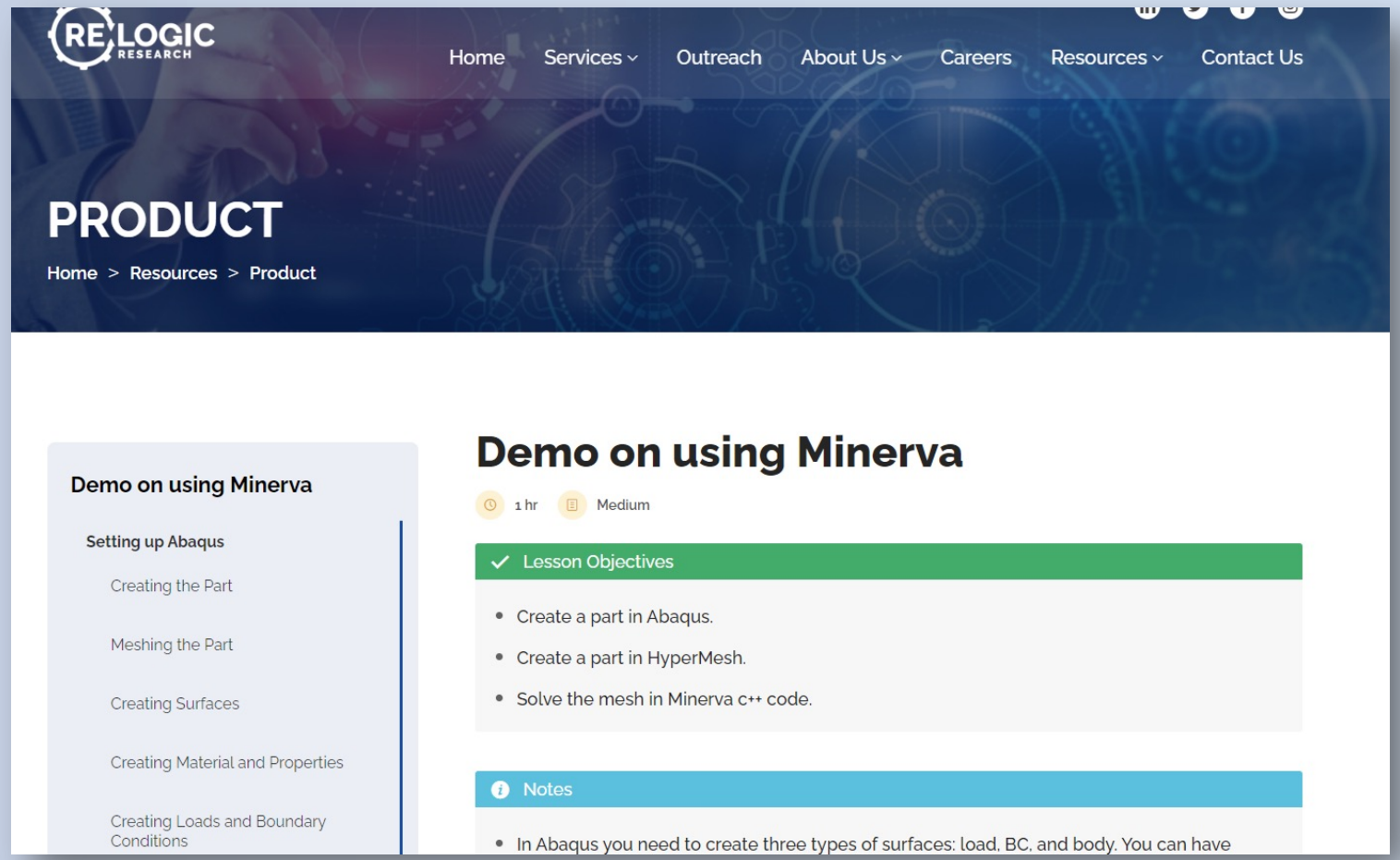


PLATFORM FEATURES

- The current layer translates an Abaqus input file into a MFEM mesh file and a MFEM model file
  - Abaqus/CAE
    Hypermesh
  - Femap (by end of 2023)
- The layer currently supports:
  - 3D solid continuum elements (tet, hex)
  - Essential BCs (surfaces not nodes)
  - Surface loads (pressure, traction, etc.)
  - Multiple isotropic materials definitions
  - Static, linear analysis
- The MFEM mesh file (*.mesh) is generated completely by the layer
- The MFEM model file (*.cpp) is generated by automatically populating fields in a template for static analysis
  - Serial
  - Parallel
  - Parallel + AMR

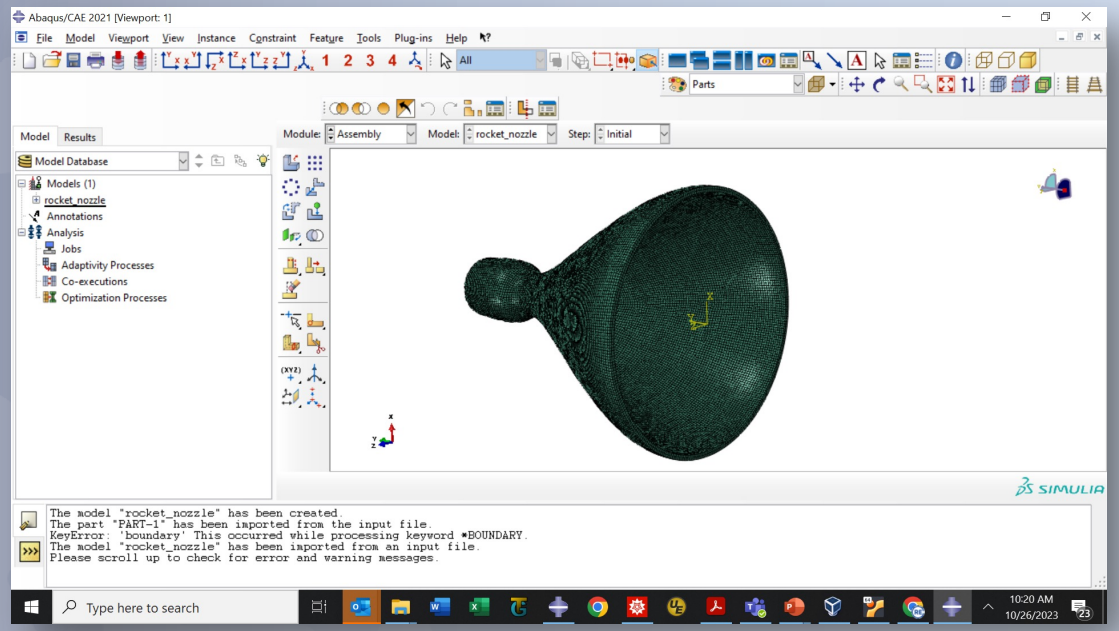
TUTORIALS

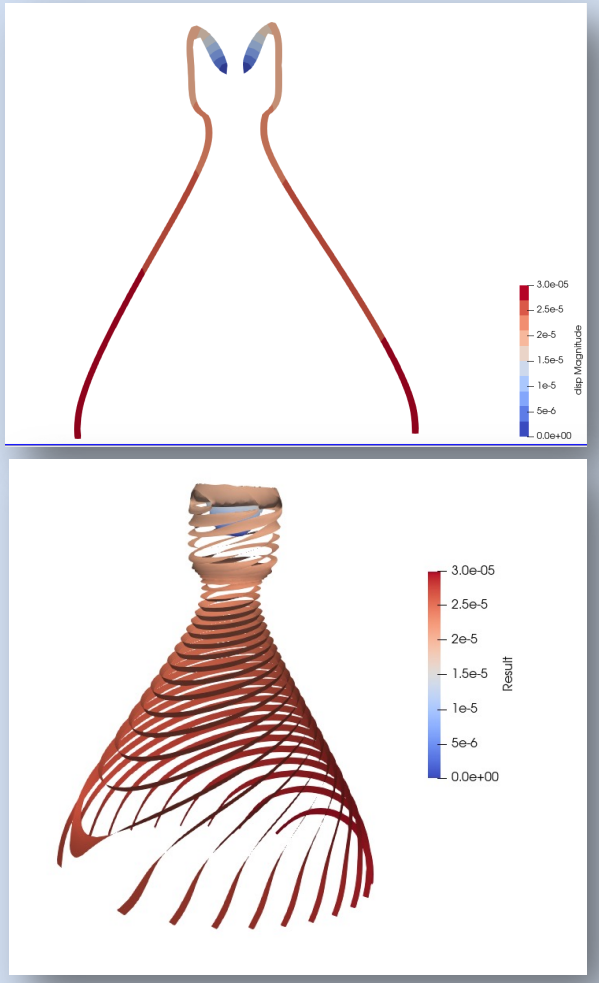
- Available on ReLogic's webpage
- Examples for creating models in:
  - Abaqus/CAE
  - Hypermesh
  - Femap
- Examples on using Minerva to generate MFEM analysis files
- Examples of post-processing in:
  - Paraview
  - Hypermesh
  - Femap



EXAMPLE

- Simple rocket nozzle with internal pressure





DEPLOYMENT

- Tutorials
- Web based front-end
- Coming to the MFEM Community by the end of 2023!