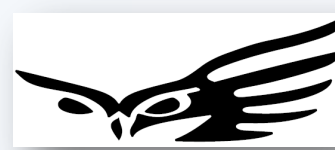


# PROJECT MINERVA

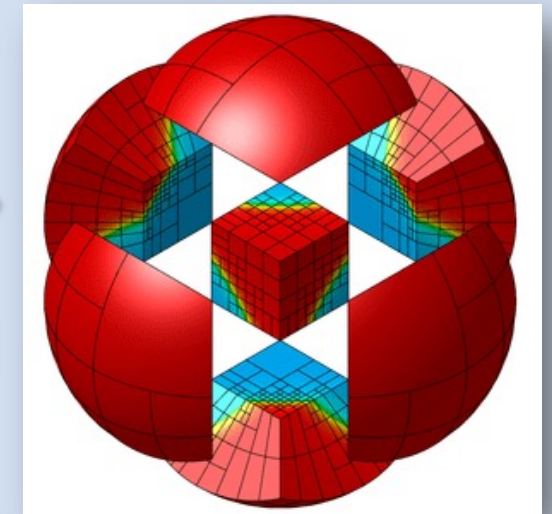
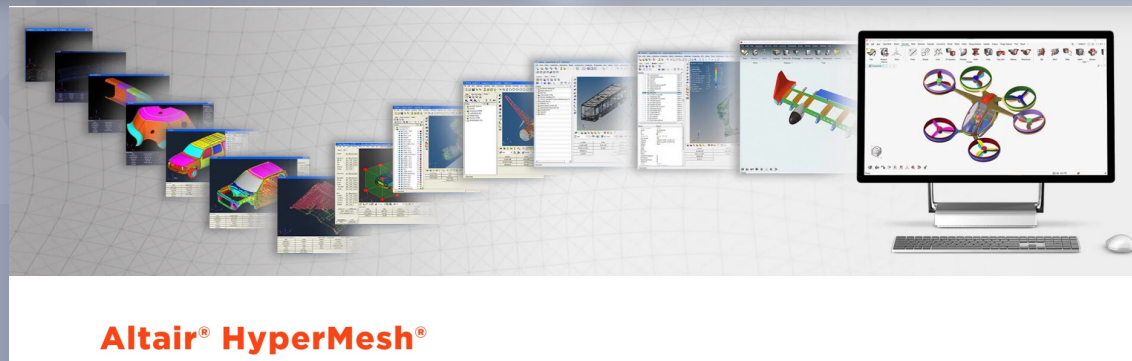
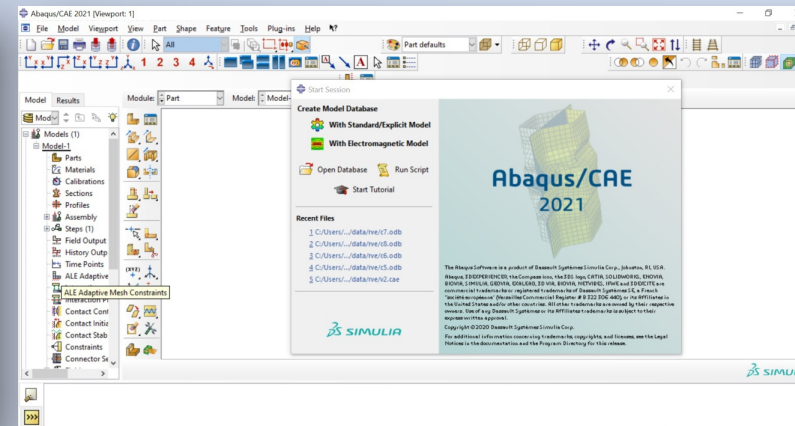
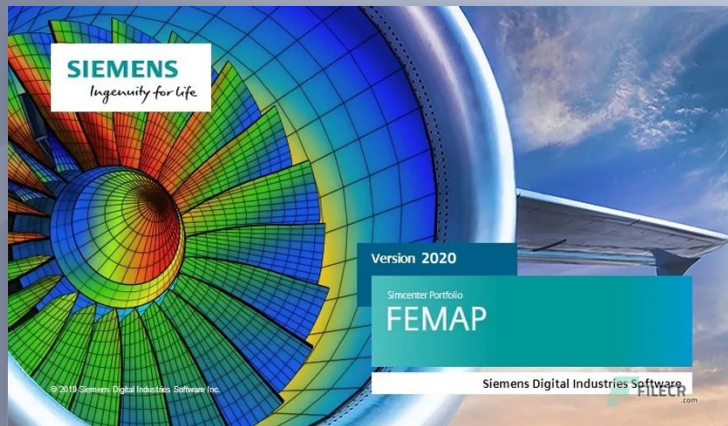
Accelerated Deployment of MFEM Based Solvers in  
Large Scale Industrial Problems Topic: 2/a



# INTRO



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

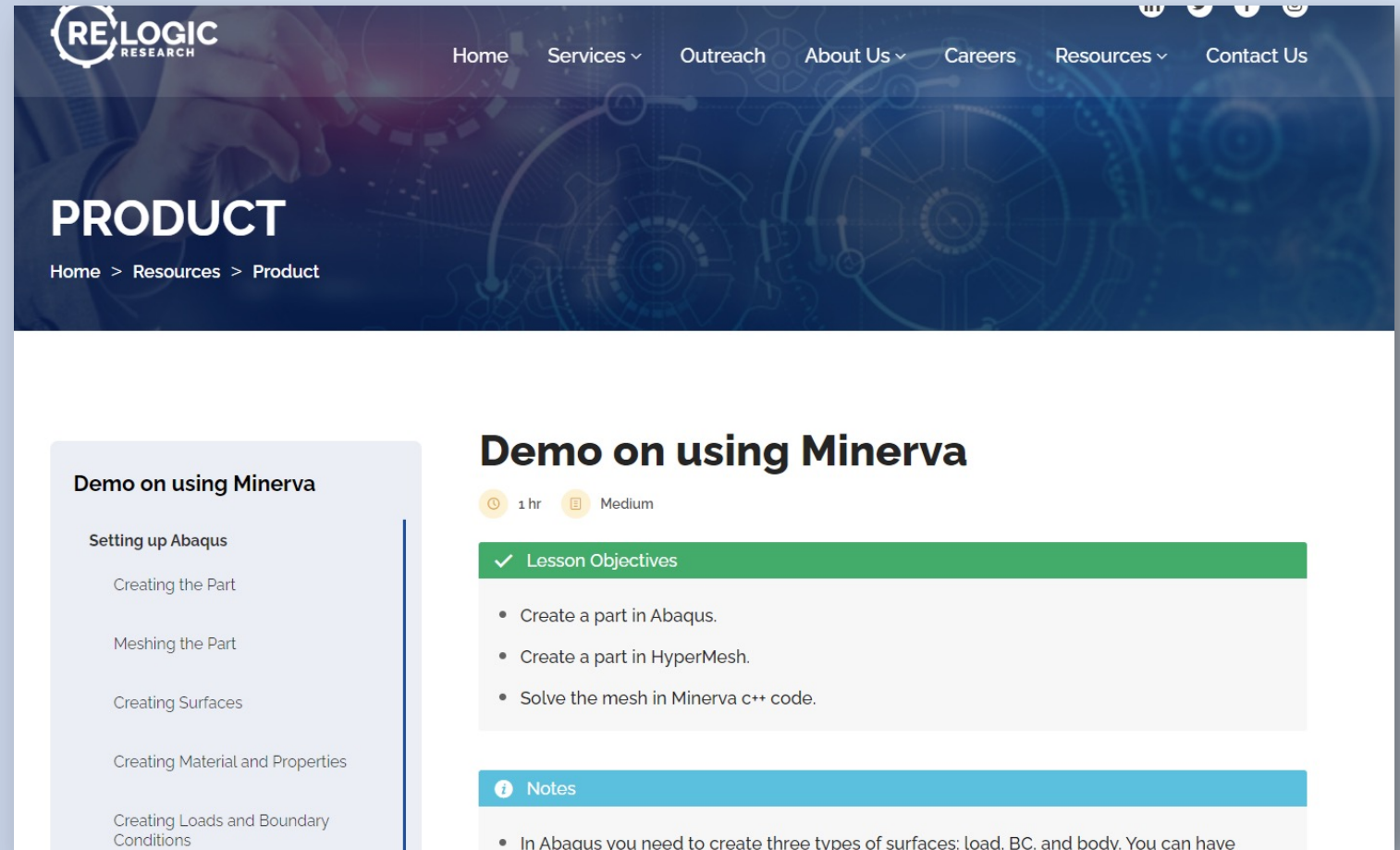


# PLATFORM FEATURES



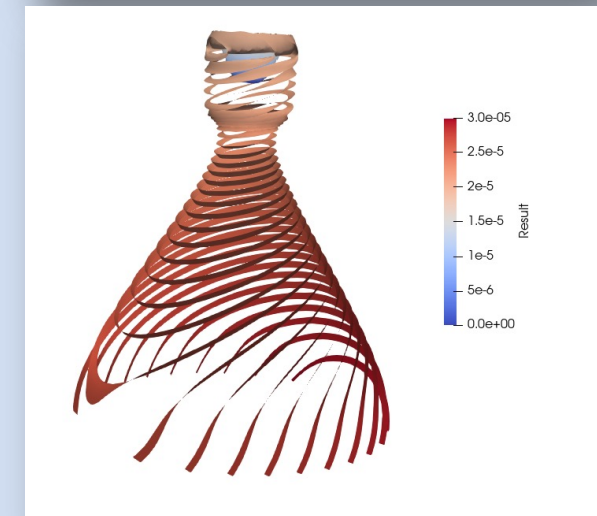
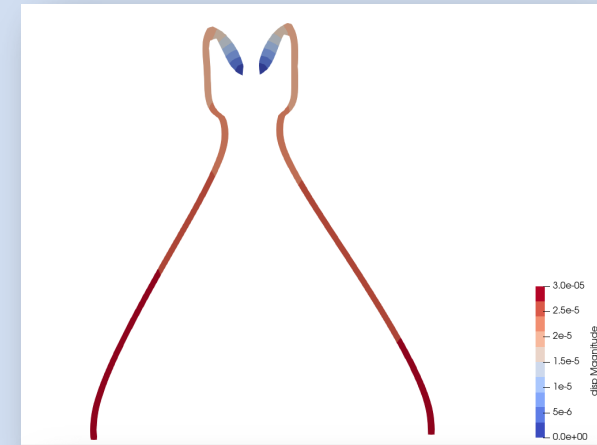
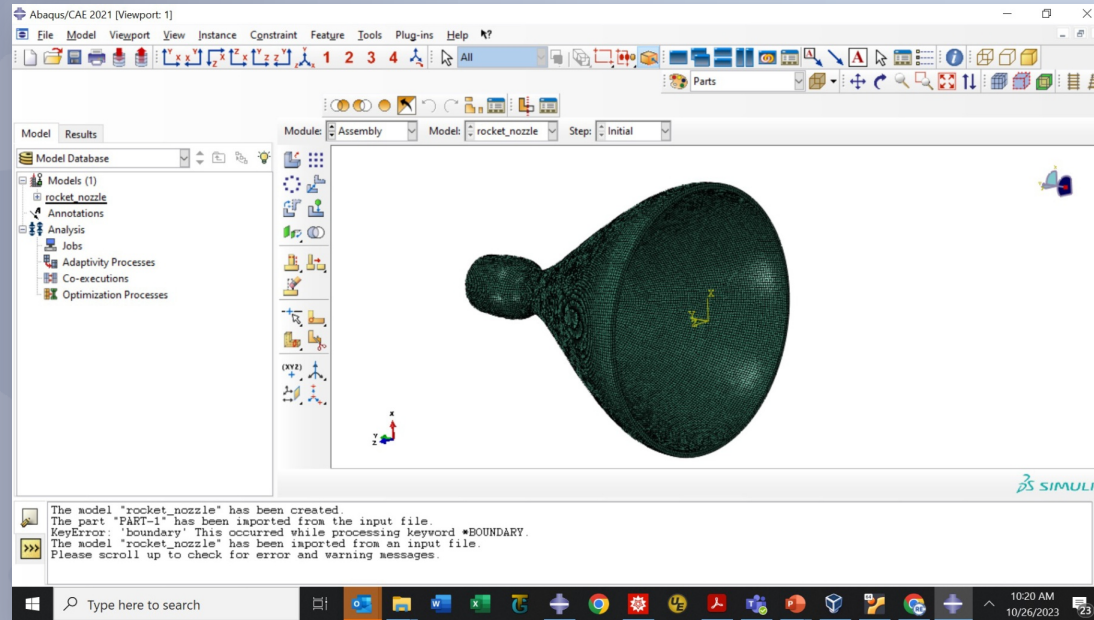
- The current layer translates an Abaqus input file into a MFEM mesh file and a MFEM model file
  - Abaqus/CAE  
Hypermesh
  - Femap (by end of 2023)
- The layer currently supports:
  - 3D solid continuum elements (tet, hex)
  - Essential BCs (surfaces not nodes)
  - Surface loads (pressure, traction, 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

- 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

A screenshot of the ReLogic Research website. The top navigation bar includes the ReLogic Research logo and links for Home, Services, Outreach, About Us, Careers, Resources, and Contact Us. The main content area is titled "PRODUCT" and includes a breadcrumb trail: Home > Resources > Product. Below this, there are two columns. The left column is titled "Demo on using Minerva" and lists several steps: Setting up Abaqus, Creating the Part, Meshing the Part, Creating Surfaces, Creating Material and Properties, and Creating Loads and Boundary Conditions. The right column is titled "Demo on using Minerva" and shows a lesson objective: "Lesson Objectives" with a green checkmark. Below this, there are three bullet points: "Create a part in Abaqus.", "Create a part in HyperMesh.", and "Solve the mesh in Minerva c++ code." There is also a "Notes" section with a blue header and one bullet point: "In Abaqus you need to create three types of surfaces: load, BC, and body. You can have".

# EXAMPLE

- Simple rocket nozzle with internal pressure




# DEPLOYMENT



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

## MFEM Mesh Conversion Tool

Developed By [ReLogic Research](#)



---

**Instructions:**

To use the tool the user must enter the contact information provided in the fields below. Input files uploaded to the tool must be in X, Y, or Z input formats. Click the "Run Application" button to process the file. The converted files will be archived into a .zip archive that can be accessed by clicking the "Download Output" button. Any errors that are generated will be stored in the .log file in the .zip archive.

**Disclaimer:**

ReLogic will store contact information for marketing purposes, but will not share with any third party entities. All files uploaded to the server are deleted after the user session is closed and will not be accessed or used by ReLogic.

**User Agreement:**

...

**Developer Contact Information:**

If you have questions or issues using this tool please contact John.Doe@relogic.com or at (234) 567-8910.

**User Information:**

<input type="text" value="Enter First Name"/>	<input type="text" value="Enter Last Name"/>
<input type="text" value="Enter Email"/>	<input type="text" value="Enter Company"/>

**Convert Files:**

Drag and Drop or Select File

---

**Output:**