CSC

Elmer Alternative Pre-processing tools

ElmerTeam CSC – IT Center for Science

Mesh generation capabilities of Elmer suite

CSC

ElmerGrid

native generation of simple structured meshes

ElmerGUI

- plugins for tetgen, netgen and ElmerGrid
- No geometry generation tools to speak about
- No capability for multibody Delaunay meshing
- Limited control over mesh quality and density
- Complex meshes must be created by other tools!

Open Source software for Computational Engineering

Code Aster

















Visualization Toolkit

CODE SATURNE



BMA

Code less. Create more. Deploy everywhere.

python

Open source software in computational engineering

- Academicly rooted stuff is top notch
 - Linear algebra, solver libraries
 - PetSc, Trilinos, OpenFOAM, LibMesh++, ...
- CAD and mesh generation not that competitive
 - OpenCASCADE legacy software
 - Mesh generators netgen, tetgen, Gmsh are clearly academic
 - Also for OpenFOAM there is development of commercial preprocessing tools
- Users may need to build their own workflows from the most suitable tools
 - Also in combination with commerial software

Open Source Mesh Generation Software for Elmer

- ElmerGrid: native to Elmer
 - Simple structured mesh generation
 - Simple mesh manipulation
 - Usable via ElmerGUI
- ElmerMesh2D
 - Obsolite 2D Delaunay mesh generator usable via the old ElmerFront
- Netgen
 - Can write linear meshes in Elmer format
 - Usable also as ElmerGUI plug-in
- Tetgen
 - Usable as ElmerGUI plug-in
- Gmsh
 - Includes geometry definition tools
 - ElmerGUI/ElmerGrid can read the format msh format
- SALOME
 - ElmerGrid can read the unv format written by SALOME
- Triangle
 - 2D Delaunay
 - ElmerGUI/ElmerGrid can read the format

Commercial mesh generation software for Elmer

- GiD
 - Relatively inexpensive
 - With an add-on module can directly write Elmer format

- Comsol multiphysics
 - ElmerGUI/ElmerGrid can read .mphtxt format

- Ask for your format:
 - Writing a parser from ascii-mesh file usually not big a deal

Mesh generation tools – Poll (5/2017)

What mesh generation software do you use with Elmer?

ElmerGUI (netgen or tetgen plugins)	10	9%
Gmsh	46	43%
Netgen	11	10%
ElmerGrid (native .grd format)	9	8%
GiD	1	1%
Ansys	3	3%
Gambit	0	No votes
Comsol Multiphysics	1	1%
Salome	22	20%
Something else (please specify)	5	5%
	Total votes : 108	

CAD – SALOME

http://www.salome-platform.org/

- SALOME is an open-source software that provides a generic platform for Pre- and Post-Processing for numerical simulation. It is based on an open and flexible architecture made of reusable components.
- SALOME is a cross-platform solution. It is distributed as open-source software under the terms of the GNU LGPL license. You can download both the source code and the executables from this site.
- SALOME can be used as standalone application for, or as a platform for integration of the external thirdparty numerical codes.





Using Salome with Elmer

There are some instructions in Wiki

http://www.elmerfem.org/wiki/index.php/Salome

C S C

- The .unv format provides a channel from Salome to Elmer
 - ElmerGrid 8 2 test.unv –autoclean
 - Or direct opening with ElmerGUI
- Unv import of ElmerGrid tries to maintain the names and save them to mesh.names file of mesh directory
 - Set "Use Mesh Names = True" to Simulation section
- There is active development of Elmer plug-in by the open source community
 - Follow discussion on the Elmer forum

Gmsh

http://gmsh.info

- Written by Christophe Geuzaine and Jean-François Remacle
- Gmsh is a free 3D finite element grid generator with a build-in CAD engine and post-processor

- Its design goal is to provide a fast, light and user-friendly meshing tool with parametric input
- Gmsh is built around four modules: geometry, mesh, solver and post-processing.
- The specification of any input to these modules is done either interactively using the graphical user interface or in ASCII text files using Gmsh's own scripting language.
- Probably the most popular academic mesh generation for finite element method

Using Gmsh with Elmer

- Saving of the mesh in native gmsh format
 - Suffix .msh
- Usually saving all geometric entities is most robust method
 - Elmer automatically drops lower dimensional entities
 - Elmer renumbers BCs and bodies with 1,2,3,....
- In practice:
- In Gmsh:
 - File -> Save as Filename: test.msh MSH Options Version 2.0 ASCII Save all (ignore physical groups)
- In ElmerGUI
 File -> Open : test.msh
- Or ElmerGrid:
 ElmerGrid 14 2 test.msh -autoclean (creates a mesh file in directory test)



Example: exporting tutorial 2 of Gmsh





Exercise: Gmsh to Elmer export

- Start gmsh.exe
- Load a existing tutorial in Gmsh

— t1-t6

- Create the default mesh for it
 - Mesh -> 1D, 2D, (3D)
 - A global size factor may be found at
 - Options Mesh General Max. Element size
- Open the mesh in ElmerGUI
- Perform a simple thermal analysis if you have time



Tutorial 2 of Gmsh

Netgen

http://www.hpfem.jku.at/netgen/

- Developed mainly by Joachim Schöberl
- An automatic 2D/3D tetrahedral mesh generator
- Accepts input from constructive solid geometry (CSG) or boundary representation (BRep) from STL file format
- Connection to OpenCASCADE deals with IGES and STEP files
- Modules for mesh optimization and mesh refinement
- LGPL library



- Netgen as a library is utilized by a large number of GUI projects
- Directly writes meshes in Elmer format (linear only)

GiD

http://www.gidhome.com

- GiD is devoped at CIMNE, Barcelona
- GiD is a universal, adaptive and user-friendly pre and postprocessor for numerical simulations in science and engineering.
- Designed to cover all the common needs in the numerical simulations field from pre to post-processing: geometrical modeling, effective definition of analysis data, meshing, data transfer to analysis software, as well as the visualization of numerical results.
- A good compromise between features and price
- Enables creation of hybrid meshes (not well supported in Gmsh)
- Elmer plugin for writing meshes in Elmer exist

Using GID with Elmer

- Requires special plugins that enable problemtype "Elmer"
- Saves Elmer mesh files directly
- For more details see: http://www.csc.fi/english/ pages/elmer/interfaces



Summary of Pre-Processing Workflows in Elmer

- Simple structured
 - ElmerGrid -> ElmerSolver
- Intermediate academic
 - Gmsh -> ElmerGrid/ElmerGUI -> ElmerSolver
- Complex free
 - SALOME -> ElmerGrid -> ElmerSolver
- Complex commercial
 - GiD -> ElmerSolver



CSC

Elmer Post-processing utilities

ElmerTeam CSC – IT Center for Science

Visualization capabilities of Elmer suite

ElmerPost was basically ok but had some limitations

- Somewhat outdated look and feel
- Output resolution same as window resolution
- Only one view at a time
- No parallel functionality
- Some compilation challenges
- VTK-widget in ElmerGUI
 - Minimalistic visualization mimicing ElmerPost functionality
 - Nice as an integrated tool for educational purposes
 - Not actively developed
- Visualization tools beyond Elmer suite as mainly used
 - Tools based on VTK library!

Visualization tools – Poll (5/2017)

What visualization software do you use?

ElmerPost	14	16%
ElmerGUI VTK postprocessor	9	11%
Paraview	37	44%
ViSit	3	4%
Mayavi	0	No votes
Gmsh	4	5%
GiD	1	1%
Matlab	7	8%
gnuplot	4	5%
Something else (please specify)	6	7%
	Total votes : 85	

Exporting FEM data: ResultOutputSolve

- Apart from saving the results in .ep format it is possible to use other postprocessing tools
- ResultOutputSolve offers several formats
 - vtk: Visualization tookit legacy format
 - vtu: Visualization tookit XML format
 - Gid: GiD software from CIMNE: http://gid.cimne.upc.es

- Gmsh: Gmsh software: http://www.geuz.org/gmsh
- Dx: OpenDx software
- Vtu is the recommended format!
 - offers parallel data handling capabilities
 - Has binary and single precision formats for saving disk space
 - Suffix .vtu in Post File does this automatically

Exporting 2D/3D data: ResultOutputSolve



An example shows how to save data in unstructured XML VTK (.vtu) files to directory "results" in single precision binary format.

```
Solver n
Exec Solver = after timestep
Equation = "result output"
Procedure = "ResultOutputSolve" "ResultOutputSolver"
Output File Name = "case"
Output Format = String "vtu"
Binary Output = True
Single Precision = True
End
```

Basic functionality also just by adding suffix **.vtu** to the **Post File** in simulation section

ParaView

http://www.paraview.org

- Developed by Kitware and US national labs (Los Alamos, Sandia, etc.)
- ParaView is an open-source, multi-platform data analysis and visualization application based on VTK
- Data exploration can be done interactively in 3D or programmatically using ParaView's batch processing capabilities.
- ParaView was developed to analyze extremely large datasets using distributed memory computing resources. It can be run both on supercomputers and laptops.
- Most popular OS visualization tool for FEM data

ViSiT



https://visit.llnl.gov/

- Devloped at Lawer Livermore National Labs.
- Visit is an open source, interactive, scalable, visualization, animation and analysis tool.
- From Unix, Windows or Mac workstations, users can interactively visualize and analyze data from small desktop projects to huge HPC projects
- Visit contains a rich set of visualization features to enable users to view a wide variety 2D and 3D data, structured and un-structured meshes

Comparison of visualization software

ParaView

- Fulfils the standard needs for FEM simulation
- Supports Elmer best via the VTU file format
- Look and feel is very appealing
- Filters are applied directly after they have been selected
 - Interactive operation nice with small datasets
- Good 1st choice

Vislt

- Fulfils the standard needs for FEM simulation
- Supports Elmer best via the VTU file format
- Look and feel may feel somehwat academic
- Whole workflow is applied only after request
 - Enables the software to better optimize the rendereing process
- Choice for powerusers?



Case: View in Paraview



Example: view in GiD



Example: view in Gmsh



-

[0] 🕨 [1] 🕨

[2] 🕨

[3] 🕨



Visualization with Paraview



Exporting 2D/3D data: ResultOutputSolve

By setting suffix for **Post File** to **.vtu** paraview format is saved automatically.

An example shows how to save data in unstructured XML VTK (.vtu) files to directory "results" in single precision binary format.

```
Solver n
Exec Solver = after timestep
Equation = "result output"
Procedure = "ResultOutputSolve" "ResultOutputSolver"
Output File Name = "case"
Output Format = String "vtu"
Binary Output = True
Single Precision = True
Save Geometry Ids = True
End
```

Filename conventions

- Suffix of unstructured XML based VTU file is .vtu
- Timesteps numbered #step
- Partitions numbered with #partpar#step
- Holder for vtu files in parallel is .pvtu



Loading data

/// ParaView 3.14.1 64-bit			
File Edit View Sources Filters Tools Macros Help			
	G		
	1000		
Pipeline Browser /// Open File: (open multiple files with < ctrl> key.)			
■ builtin: Look in: C:/elmerwrk/Viz/			
Properties Display Properties Display File name: casevtu0001.vtu File name: casevtu0001.vtu OK File name: casevtu0001.vtu OK File name: Casevtu0001.vtu Cancel			

CSC

Note: Paraview may have several datasets at the same time!

Solid color



Moving object in Paraview

csc

- Rotate
 - Mouse: Left bottom
- Scale
 - Mouse: Right bottom
- Translate
 - Mouse: Center bottom

Setting background color



Color mesh with surface + edges


AMR Contour

AMR Dual Clip

- Annotate Time Filter
- Append Attributes

Append Datasets Append Geometry

Block Scalars

Calculator Cell Centers

Cell Data to Point Data Clean

Clean Cells to Grid

Clean to Grid Clip

Clip Closed Surface

Clip Generic Dataset

- Compute Derivatives
- . Connectivity
- Contingency Statistics
- Contour
- Contour Generic Dataset

Curvature

D3

Decimate

Delaunay 2D

Delaunay 3D

Descriptive Statistics

Elevation

Extract AMR Blocks

Extract Block

- Extract CTH Parts
- Extract Cells By Region

Extract Edges Extract Generic Dataset Surface

Extract Level

Extract Selection

Extract Subset

- Extract Surface
- FFT Of Selection Over Time FOF/SOD Halo Finder

Feature Edges

Gaussian Resampling

Generate Ids

Generate Quadrature Points

Generate Quadrature Scheme Dictionary

Generate Surface Normals

Glyph Glyph With Custom Source Gradient Gradient Of Unstructured DataSet Grid Connectivity

Group Datasets Histogram

- Image Data to Point Set
- Integrate Variables Interpolate to Quadrature Points

Intersect Fragments Iso Volume

K Means Level Scalars

Linear Extrusion

- Loop Subdivision
- Mask Points
- Material Interface Filter
- Median
- Merge Blocks Mesh Quality

Multicorrelative Statistics

- Normal Glyphs Octree Depth Limit Octree Depth Scalars
 - Outline Outline Corners
 - Outline Curvilinear DataSet

Particle Pathlines ParticleTracer

Plot Data

- Plot Global Variables Over Time
- Plot On Intersection Curves Plot On Sorted Lines
- 💉 🛛 Plot Over Line
- Plot Selection Over Time Point Data to Cell Data
 - Principal Component Analysis Probe Location
 - Process Id Scalars
- {...} Programmable Filter Python Calculator Quadric Clustering

Random Vectors Rectilinear Data to Point Set

Rectilinear Grid Connectivity Reflect

Resample With Dataset

Ribbon Rotational Extrusion Scatter Plot Shrink

Slice Slice Generic Dataset

Smooth Stream Tracer

- Stream Tracer For Generic Datasets Stream Tracer With Custom Source Subdivide Surface Flow Surface Vectors Table To Points Table To Structured Grid Temporal Cache Temporal Interpolator Temporal Shift Scale Temporal Shift Scale Temporal Statistics Tessellate Tetrahedralize Texture Map to Cylinder
- Texture Map to Plane Texture Map to Sphere
- Threshold Transform Triangle Strips Triangulate

Tube

Warp By Scalar

Warp By Vector Youngs Material Interface builtin:
 case0001.pvtu
 Connectivity1
 Slice1
 Glyph1



 Paraview uses extensively *filters* to create new datasets

- Filters and datasets may be set active or passive by clicking the eye
- Several datasets may be visualized at the same time

Plotting a slice



Plotting a clip



Vector plot



Vector plot + opaque solid surface

/// ParaView 3.14.1 64-bit	
File Edit View Sources Filters Tools Macros Help	
🗧 🎴 🚔 💿 temperature 🔷 🔍 Surface 🗸 🔀 🖾 💥 🚔 🛱 🛱 🛱 🎜 🖉 🚱 🚱	
[1] I = 1 [2] I = 1 [2] [2] I = 1 [2]	
Pipeline Browser B × Layout #1 × +	
builtin: Caseytu0001.vtu Glyph1 Properties Display Information	
Project des Coparty Display Coparty Style Coparty Point size 2,00 Line width 1,00 Coparty 0,10 Subdivision 1 Edge Style Coparty Volume Volume Volume Projected tetra	
	h.

Vector plot + solid surface with Id treshold



Change of colormap

	C S C
M ParaView 3.14.1 64-bit	
File Edit View Sources Filters Tools Macros Help	
	0
Color Scale Color Legend	
Render View Immediately Save Choose Preset	
Pipeline Brow	
Cales Seales Value	
Color Scalar Value Invan Color Color Space Diverging Use Logarithmic Scale	
Automatically Rescale to Fit Data Range	
Minimum: 10 Maximum: 34.3139	
Rescale Range Rescale to Data Range Rescale to Temporal Range	
Use Discrete Colors	
Properties 16	
Display	
View	
Visible Apply Make Default Close	
Select	
Color	
✓ Interpolate Scalars	
Map Scalars	
Apply Texture None	
Color by	
Edit Color Map Rescale to	
Slice	
Slice Direction	

Deformation – WarpByVector filter



Plot line – PlotOverLine filter



Streamlines – Filter StreamTracer



Partitioning – Connectivity filter



CSC

File size in Paraview output

- Memory consumption of vtu-files (for Paraview) was studied in the "swiss cheese" case
- Saving just boundaries in single precision binary format may save over 90% in files size compared to full data in ascii
- With larger problem sizes the benefits are amplified

Binary output	Single Prec.	Only bound.	Bytes/ node
-	Х	-	376.0
Х	-	-	236.5
Х	Х	-	184.5
Х	-	Х	67.2
Х	Х	Х	38.5

Simulation Peter Råback, CSC, 2012.





Saving figures

·		С
/// ParaView 3.14.1 64-bit		
File Edit View Sources Filters Tools Macros Help		
😥 🤌 🐯 🙀 🔎 🔍 🛃 🧣 🤐 🍇 👯 🙀 🕸 🔤 👘 🚱 🚺 🗤 🖉		
🚦 🍡 🖛 emperature 🔹 💽 Surface 💽 🔀 😳 🗱 🏥 🏥 🛱 🛱 🛱 🎜		
🗐 🚳 🟟 🕸 🗐 🕀 😂 🧀 🛞	1	
Pipeline Browser		
III Save Screenshot: ? 💌		
✓ Look in: C:/elmerwrk/Viz/ ✓ C C C:/elmerwrk/Viz/		
My Docun A Filename	8	
Desktop		
Jacobia Favorites	10	
Di 🔰 Viz	E Contraction of the second seco	
File name: case.png OK		
Files of type: PNG image (*.png) Cancel		
Hies of type: His mage (.prg)		
Map Scalars	8	
Apply Texture None		
Color by		
Edit Color Map Rescale to		
Slice		
Slice Direction		
Slice 0 -		
	at a	
	The second loss of the	

Saving animations with Paraview

The only packing method that comes with Paraview by default is motion AVI

CSC

- It is advicable to save the animation as separate files
- You may use ElmerClips to make mpg animations of the separate png figures



Visualization with ElmerPost

How to write files for ElmerPost

- Default suffix is .ep
- May be requested in Simulation section Post File = case.ep
- Or using ResultOutputSolver with
 Output format = ElmerPost



Loading data

- Assume data in case.ep
- File -> Open ->
 case.ep
- Here the timesteps are chosen
- If element edges or sides are not defined for BCs they may have to be created here

CSC	
🏸 Read Model File	
Status: Header Read	
Options:	
Generate Surface Element Sides	
🔲 Generate Volume Element Sides	
🔲 Generate Volume Element Edges	
File Information:	
Nodes: 11949 Elements: 69792 Timestps: 2 DOFS: 5 Vector: Velocity Scalar: Pressure Scalar: Temperature	
Select timesteps:	
First: 1 Last: 1 Increment 1 All	
Select file:	
Model file: C:/elmerwrk/Viz/case.ep Browse	
Read header Read file OK Close	



Moving object in ElmerPost



- Rotate
 - Mouse: Right bottom
 - Click:
 - Command line, e.g.: rotate 30 45 60
- Scale
 - Mouse: Both bottoms
 - Click: 🕁 🕁
 - Command line: scale 1 10 1
- Translate
 - Mouse: Left bottom
 - Click: ◄◄¥♠
 - Command line: translate 1 2 3

Setting background color

CSC

Olick:

- Edit -> Background
- Set 100.0 100.0 100.0 for white
- Command line
 - background 100 100 100



Color mesh with surface + edges



CSC

Plotting isosurfaces



CSC

Using clip planes



% clip_edit	
Low X Plane: -1.250	-1.250
High X Plane: 0.950	0.950
Low Y Plane: 1.250	-1.250
High Y Plane: 1.250	1.250
Low Z Plane: 1.250	-1.250
High Z Plane: 0.950	0.950
Apply Of	Close



Isosurface + surface plot + clip planes



Vector plots

ELMER POST GRAPHICS	7% vector
	Vector Length Scale: 1.00 Line Style: Line © Solid Line Quality: 1 Width Scale: 1 Threshold Variable: none Min: 0.0 Max: 1.00 Color Variable: Velocity_abs Length Variable: Velocity_abs Arrow Variable: Velocity Apply Close

C S C

Vector plot + solid surface



Surface plot + Isosurfaces + Op	aque	
ELMER POST GRAPHICS	x	76 Material
		Ambient & Diffuse C Specular
		Shininess
		25.0 0.0 32.0 64.0 96.0 128.0
	1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1	Opacity (%)
	a statistic	30.0
	7∦ isosurface	90.0
	13.4734079143 16.9468158286 20.4202237428 Number Of Isosurfaces: 6	90.0
	23.8936316572 27.3670395714 30.8404474857 Min: 10.0 Max: 34.3138554	90.0 0.0 25.0 50.0 75.0 100.0
	Surface Style: C Line 📀 Surface C Both	
	Line Style:	▲ alice blue AliceBlue
	Contour Variable: Temperature Color Variable: Temperature	antique white AntiqueWhite AntiqueWhite1 AntiqueWhite2
	Min: 10 Max: 34.314 Keep	AntiqueWhite3 AntiqueWhite4
	Surface Normal Variable: none Apply Close	aquamarine aquamarine1
		Apply OK Cancel

Change of colormap



CSC

Selecting active geometric entities



Saving figures



File -> Save Image -> jpg

7% Save Screen	_ D X
Save as:	
C Postscript	
✓ Fit PS to page	
O PPM Image	
JPG Image	
Select file:	
File Name:	Browse
	Save Close



Deformation in geometry



- Assume displacement field in variable "Displacement"
- Set in command windows: math n0=nodes math nodes=n0+Displacement
- Replot

Conclusions



- Use Paraview and VTU format
- For large visualizations ViSiT could be an option



SALOME

CSC

•SALOME is an open-source software that provides a generic platform for Pre- and Post-Processing for numerical simulation.

•It can be used as standalone application for generation of CAD model, preparation for numerical calculations and post-processing of the calculation results.

•SALOME can also be used as a platform for integration of the external third-party numerical codes to produce a new application for the full life-cycle management of CAD models.

.http://www.salome-platform.org/

•SALOME GUI functions can be extended with python plugins → Elmer plugin for SALOME



CSC

Plugin developed by Rainer Jacob and Matthias Zenker

Available from GitHub (https://github.com/physici/ElmerSalomeModule)

Requirements

Elmer 8.2 or 8.3

Salome 7.8 or 8.2

Installation to the optional directory

- 1. Create a plugin directory in the root path of SALOME or somewhere convenient, if not already using one.
- 2. Copy 'ElmerSalome' directory into the plugin directory.
- 3. Copy the 'salome_plugins.py' file in the plugin directory or modify the existing file.
- 4. Register the directory via the 'SALOME_PLUGINS_PATH' environmental variable.

Usage

In the 'Mesh'-module of Salome, the plugin is accessible via the 'Tools' \rightarrow 'Plugins' \rightarrow 'Elmer' submenu.

Related topic on Elmer forum

http://www.elmerfem.org/forum/viewtopic.php?f=15&t=3636&sid=f5e1f9a49bfc587144d508fc8639596e

Some Remarks about the Plugin



Current Features

The plugin mimics the ElmerGUI in the context of the Salome platform. It provides the same functionality as the "Model"-menu in the ElmerGUI, allowing the definition of equations, material, boundary and body properties as well as simulation related parameters like time stepping, output file, etc.

Additionally, it provides a function that allows writing the settings into a .sif file that can be used as input for the ElmerSolver.

Some Remarks about the Plugin

Remarks and Limitations

- Only for serial problems at the moment.
- Attempts to read a sif file generates error.
- Bodies and faces that shall be used for a simulation have to have a unique name without any blanks (e.g. 'Face 1' has to be 'Face1'). Ideally, these names are defined via the 'Group' function of SALOME. The plugin uses the 'Use Mesh Names'-options by default and ElmerGrid crops the names at the first occurrence of a blank.
- In the SALOME's geometry module, give names to elements needed in setting boundary and initial conditions, like 'wall', 'opening', etc.. In the meshing module, use 'Create Groups from Geometry' tool to create groups for boundary and initial conditions.
- Define first all boundary and initial conditions and then set them to desired boundaries with 'Properties of Selected Element' feature.

