Using Elmer with other pre- and postprocessors

D.Sc. Peter Råback
CSC - IT Center for Science, Finland

Elmer basic course
25.5.2010

Alternative mesh generators for Elmer

Open Source tools

- Mesh2D
  - 2D Delaunay
  - Writes Elmer format
  - Usable via the old ElmerFront
- ElmerGrid: native to Elmer
  - Simple structured mesh generation
  - Usable via ElmerGUI
- Tetgen, Netgen
  - Tetrahedral mesh generation
  - Usable via ElmerGUI as a plug-in
- Gmsh
  - Includes geometry definition tools
  - ElmerGUI/ElmerGrid can read the format
- Triangle
  - 2D Delaunay
  - ElmerGUI/ElmerGrid can read the format
- …

Commercial tools

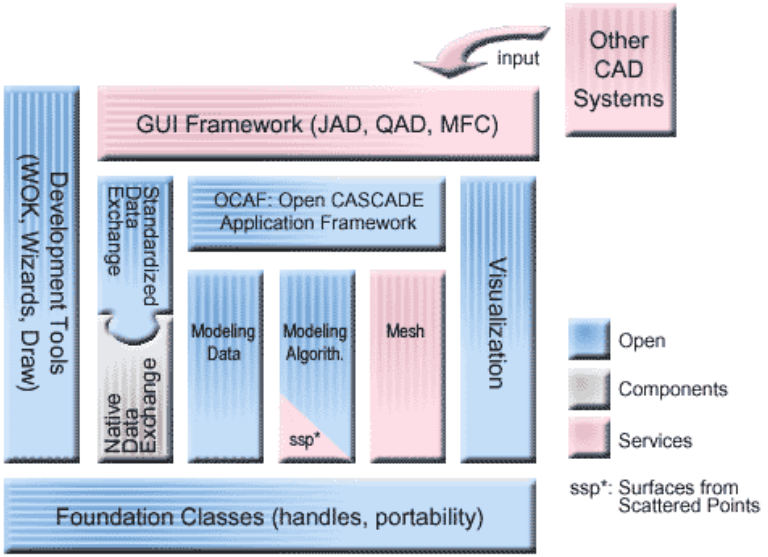
- GiD
  - Inexpensive
  - With an add-on module can directly write Elmer format
- Gambit
  - Preprocessor of Fluent suite
  - ElmerGUI/ElmerGrid can read .FDNEUT format
- Comsol multiphysics
  - ElmerGUI/ElmerGrid can read .mphtxt format
- …

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

CAD – OpenCASCADE

http://www.opencascade.com/
http://www.opencascade.org/

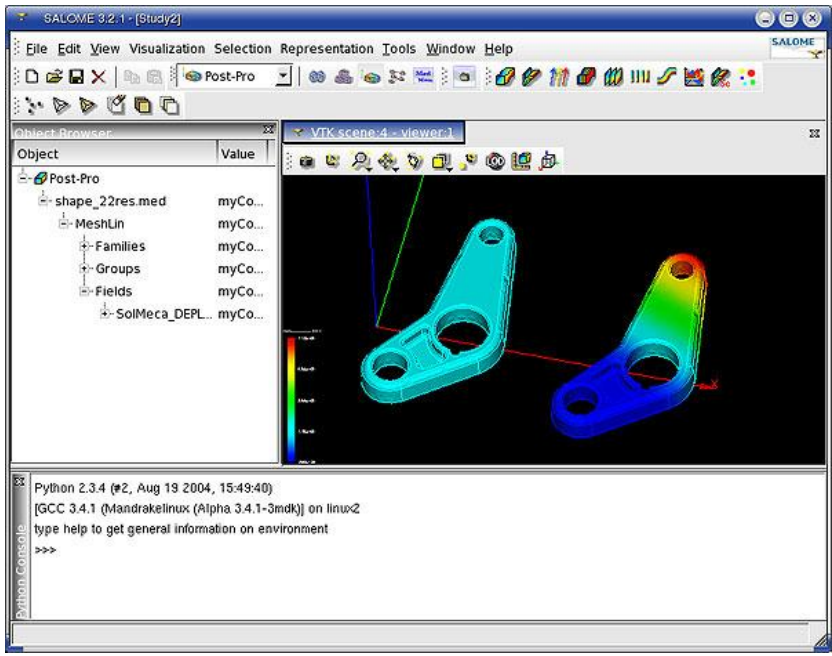
- What is it?
  - Open CASCADE is a powerful CAD/CAM/CAE kernel and development platform for 3D modeling applications.
  - It consists of reusable C++ object libraries and a set of development tools that are available in Open Source
  - Modular structure (see diagram)
- Devolopment history
  - EUCLID-IS CAD/CAM system 1987
  - Published under Open Source in 1999 as OpenCASCADE
  - OpenCASCADE S.A.S. is a service company of 80 developers
  - Curstomers CEA, BMW, SAMTECH, EADS, RINA, Alcatel,…
- The only proper CAD library under Open Source
  - Included in ElmerGUI



CAD – SALOME

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

- What is it?
  - Free software that provides a generic platform for Pre and Post-Processing for numerical simulation.
- Based on a number of free software libraries
  - Qt, OpenCASCADE, Doxygen, Python, VTK
- Main functions
  - Create/modify, import/export (IGES, STEP), repair/clean CAD models
  - Mesh CAD elements, check mesh quality, import/export mesh (MED, UNV, ASCII)
  - Handle physical properties and quantities attached to geometrical items
  - Perform computation using one or more external solvers (coupling)
  - Display computation results (scalar, vectorial)
  - Manage studies (creation, save, reload)



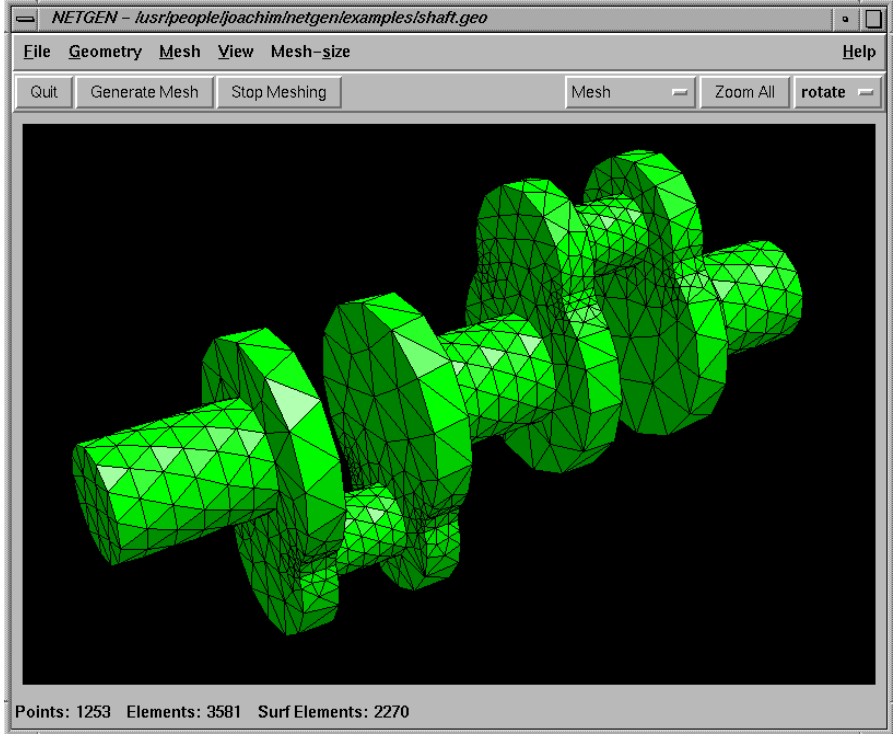
Using Salome with Elmer

There are some instructions in Wiki

- http://www.elmerfem.org/wiki/index.php/Salome
- The .unv format provides a channel from Salome to Elmer
  - ElmerGrid 8 2 test.unv –autoclean
  - Or direct opening with ElmerGUI

Meshing - Netgen

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



- What is it?
  - An automatic 2D/3D tetrahedral mesh generator
  - Developed mainly by Joachim Schöberl
- Key features
  - Accepts input from constructive solid geometry (CSG) or boundary representation (BRep) from STL file format
  - Connection to OpenCASCADE deals with IGES and STEP files
  - Contains modules for mesh optimization and hierarchical mesh refinement
  - LGPL library
- Netgen library is utilized by a large number of GUI projects

CSC

Meshing - Gmsh

http://geuz.org/gmsh/

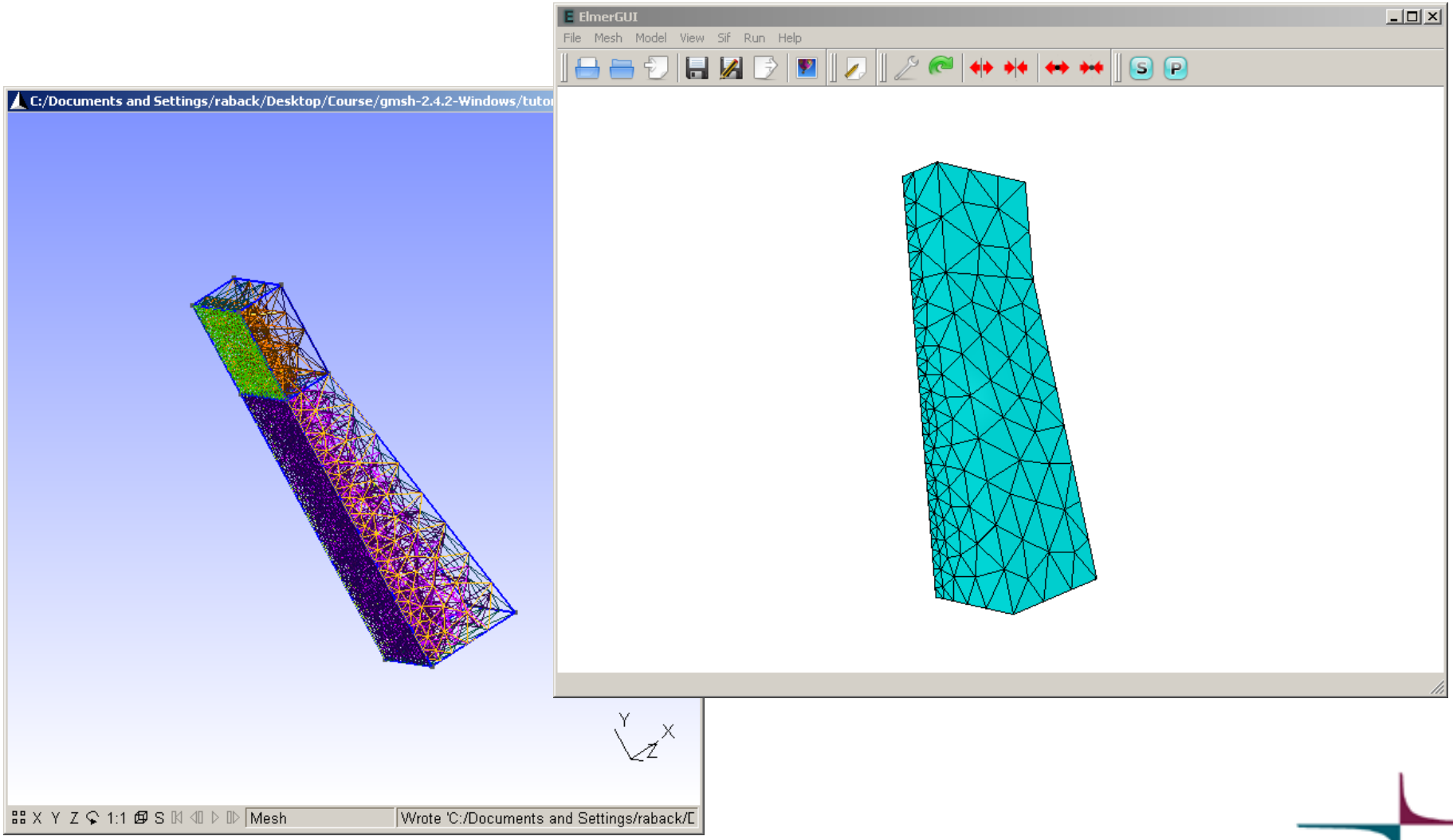
- Gmsh is a 3D finite element grid generator with a build-in CAD engine and post-processor
- Its design goal is to provide a fast, light and user-friendly meshing tool with parametric input
- Gmsh is built around four modules: geometry, mesh, solver and post-processing.
- The specification of any input 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 package under open source

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 (creates a mesh file in directory test)

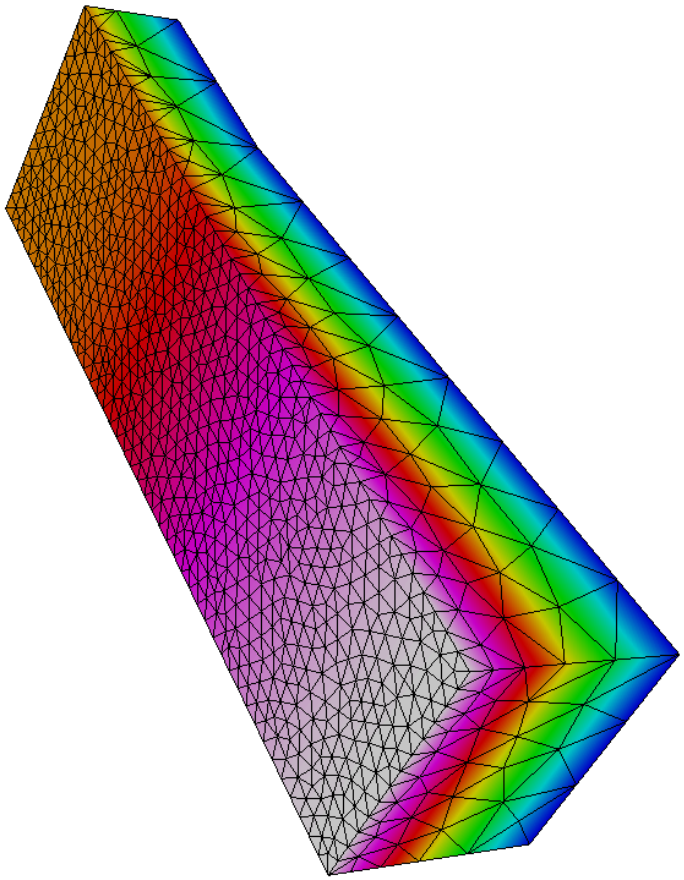
Example: exporting tutorial 2 of Gmsh



CSC

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

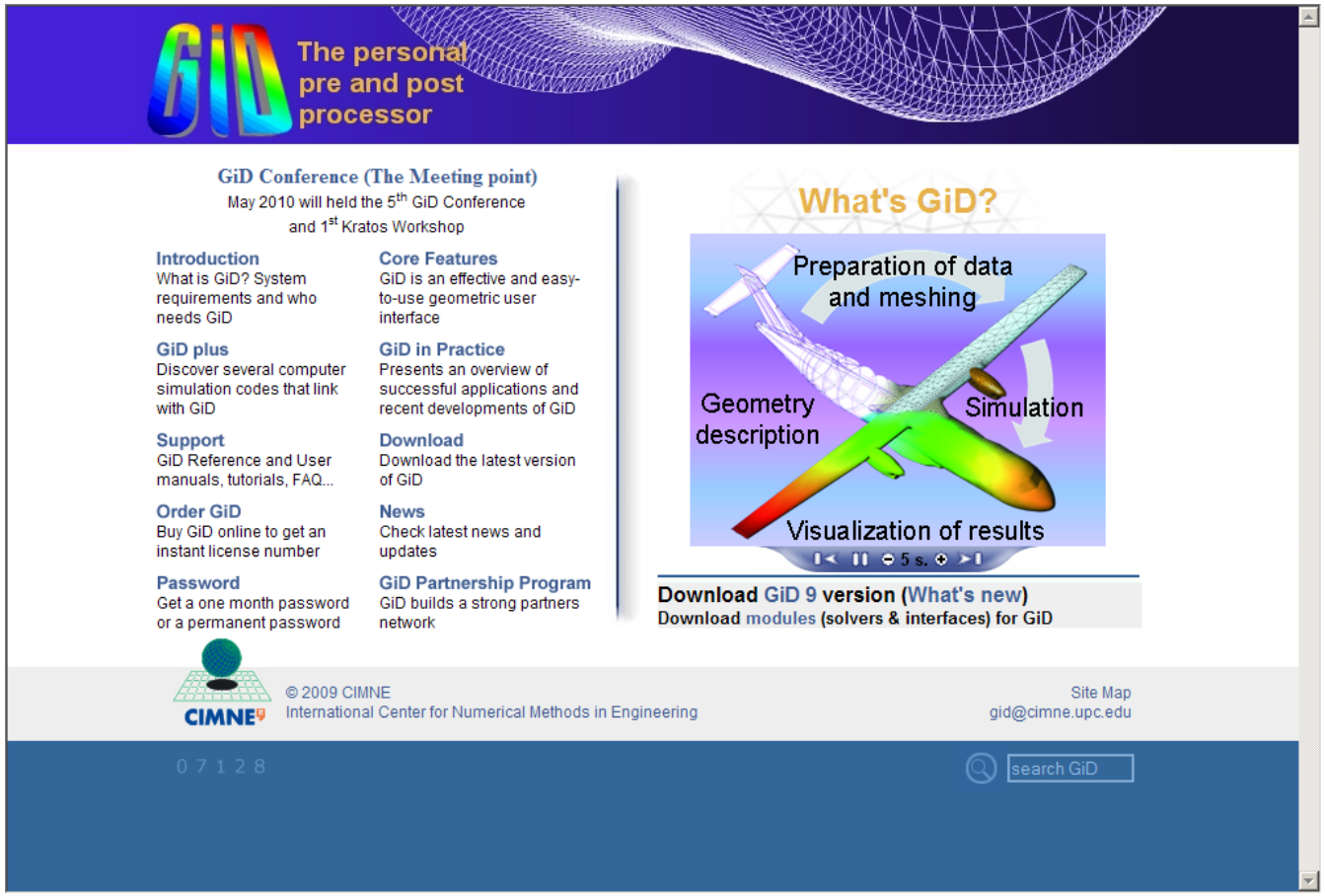


Tutorial 2 of Gmsh

GiD

http://gid.cimne.upc.es/

- A good compromise between features and price
- Enables creation of hybrid meshes (not well supported in Gmsh)



Using GID with Elmer

- Requires special plugins that enable problemtype ”Elmer”
- For more details see:
  http://www.csc.fi/english/pages/elmer/interfaces

Alternative postprocessors for Elmer

Open Source tools

- ElmerPost
  - Postprocessor of Elmer suite
- ParaView, Visit
  - Use ResultOutputSolve to write .vtu or .vtk
  - Visualization of parallel data
- OpenDX
  - Supports some basic elementtypes
- Gmsh
  - Use ResultOutputSolve to write dat
- Gnuplot, R, Octave, …
  - Use SaveData to save results in ascii matrix format
  - Line plotting

Commercial tools

- Matlab, Excel, …
  - Use SaveData to save results in ascii matrix format
  - Line plotting

Visualization - VTK

http://www.vtk.org/

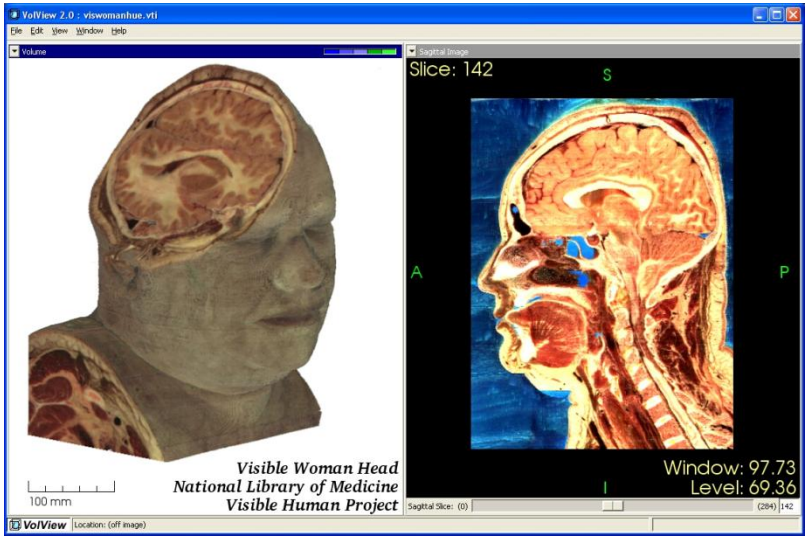
- What Is it?
  - Software system for 3D computer graphics, image processing, and visualization
- Features
  - Consists of a C++ class library and several interpreted interface layers including Tcl/Tk, Java, and Python.
  - VTK supports a wide variety of visualization algorithms including scalar, vector, tensor, texture, and volumetric methods
  - Supports parallel processing
  - Integrates with various databases on GUI toolkits such as Qt
  - VTK is cross-platform and runs on Linux, Windows, Mac and Unix platforms. An open-source, multi-platform data analysis and visualization application
- Professional support provided by Kitware Inc.
  - Proper documentation not free
  - Supported by a number of large institutions: Los Alamos an Sandia nat.lab.

VTK library is used in ElmerGUI for visualization

Visualization - VisIT

http://wci.llnl.gov/visit/

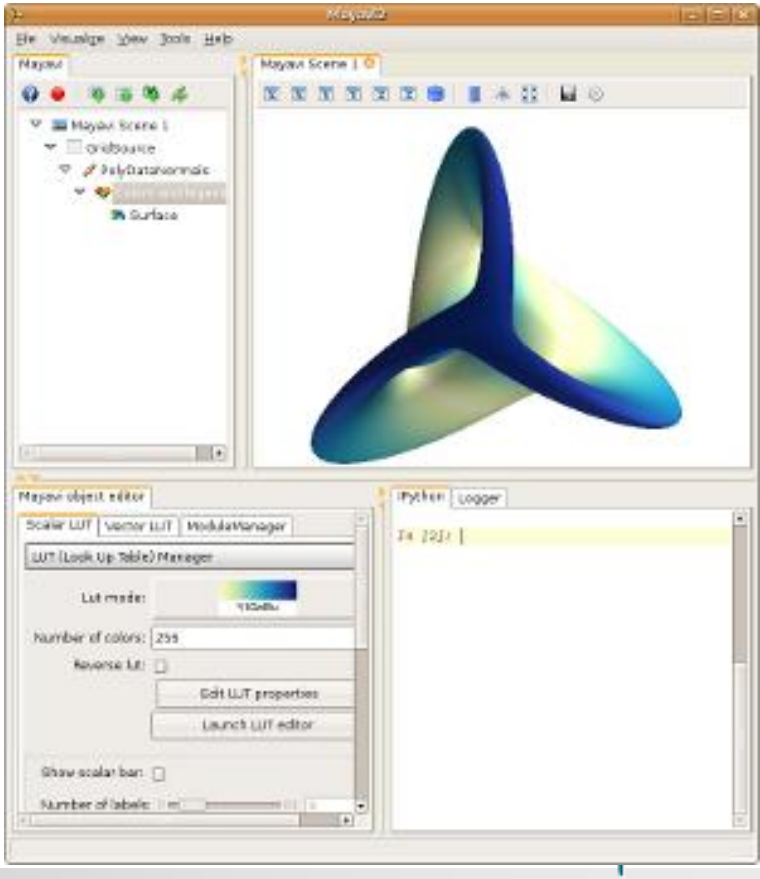
- What is it?
  - Interactive parallel visualization and graphical analysis tool for viewing scientific data on Unix and PC platforms
  - Developed by Department of Energy (DOE)
  - Rather similar in features as Paraview
  - Distribnuted under BSD license



Visualization - Mayavi

http://code.enthought.com/projects/mayavi/

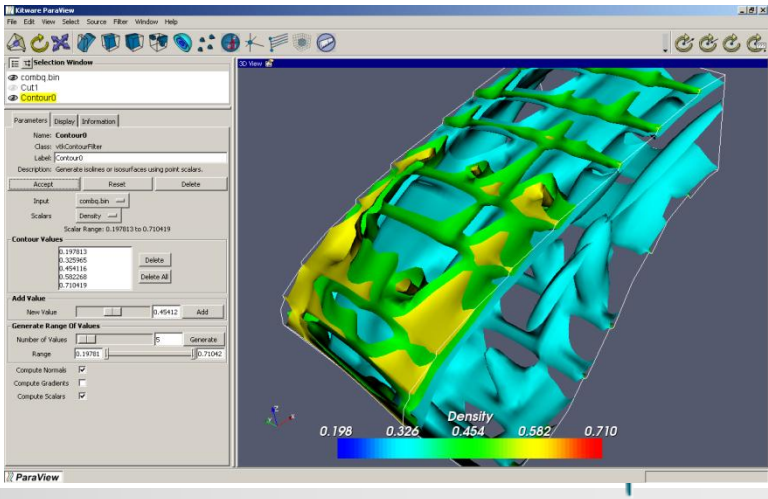
- What is it?
  - Easy and interactive visualization of 3-D data
  - Includes a (optional) rich user interface with dialogs to interact with all data and objects in the visualization.
  - A simple and clean scripting interface in Python, including one-liners, or an object-oriented programming interface.
  - Based on the VTK toolkit



Visualization - Paraview

http://www.paraview.org/

- What Is it?
  - An open-source, multi-platform data analysis and visualization application
  - Developed to analyze extremely large datasets using parallel computing
- Features
  - Data exploration may be done interactive or using batch processing
  - Can be run on laptops and supercomputers
  - Based on VTK library
- Our choice for external postprocessor



Paraview vs. ElmerPost

- Requires a separate solver in Elmer
+ Industry standard
+ Parallel files need not to be fused
+ Supports multicore visualization
+ Export of visualization separated from looking at them
+ Has a batch job mode

+ Integrated to Elmer
- Limited to Elmer
- No actual parallel support
+ May be faster for basic operations
- In exporting figures and animations the screen must be intact

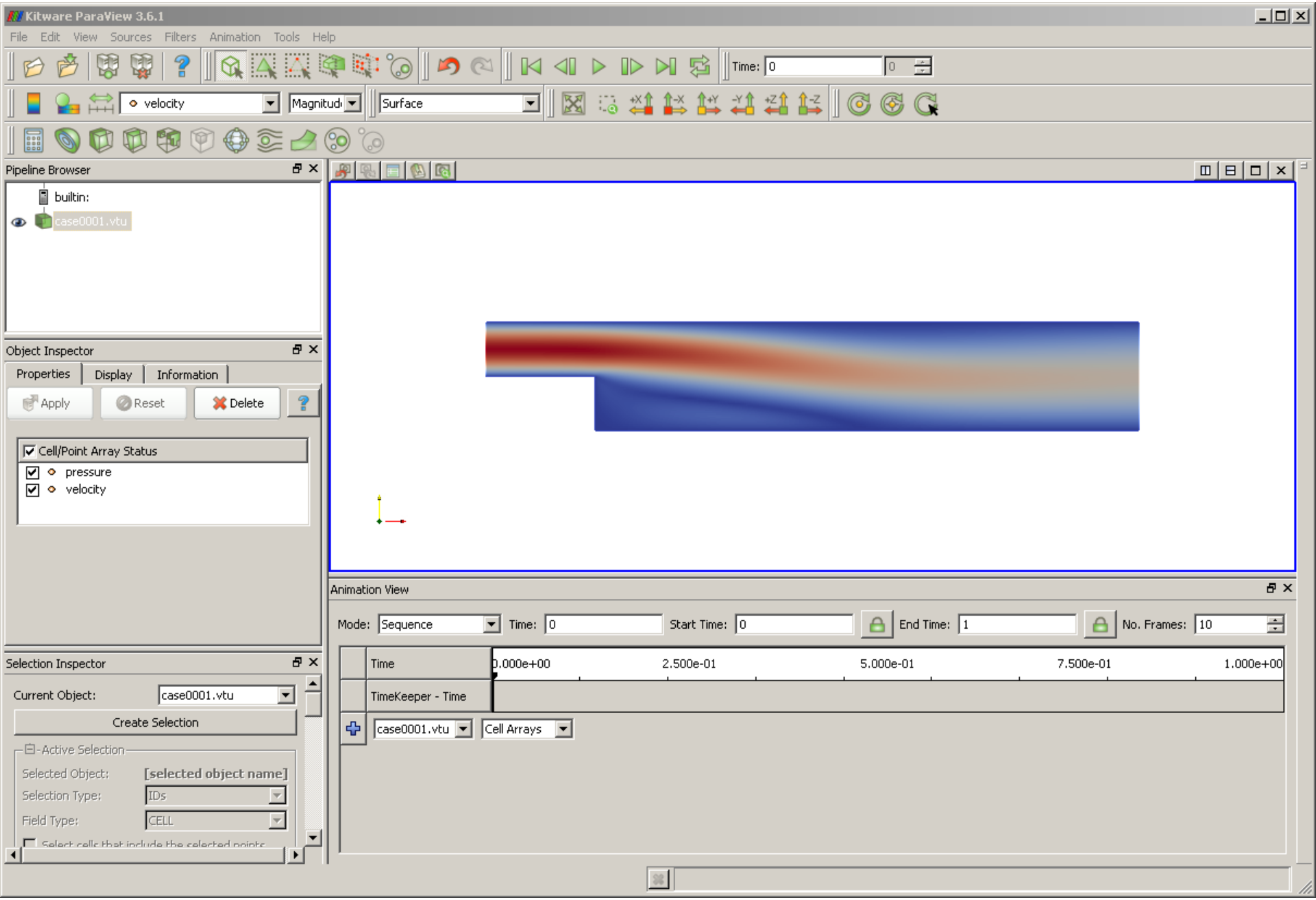
Using other postprocessors in Elmer

- The data is written by an auxiliary solver: ResultOutputSolve
- Currently supported formats include:
  - GiD
  - Gmsh
  - VTK (legacy)
  - VTU (unstructurec XML VTK)
  - DX (open DX)
- Choose by
  - Output Format = String
- GiD, VTU and Gmsh formats require explicit definition of the variables to be saved
  - Scalar Field i = String
  - Vector Field i = String
- In addition define output file name
  - Output File Name = String

Activating paraview output

- Using ElmerGUI
  - Add to the equation the solver ”Result Output” and toggle its settings in the ”Edit Solver Settings” submenu
- Manually
  - Use your favorite editor copy-paste the following settings to your sif file (with the 1st free index number for the solver)

```
Solver 2
  Equation = Result Output
  Procedure = "ResultOutputSolve" "ResultOutputSolver"
  Output File Name = case
  Output Format = Vtu
  Scalar Field 1 = Pressure
  Vector Field 1 = Velocity
End
```


Kitware ParaView 3.6.1
File Edit View Sources Filters Animation Tools Help
Time: 0
velocity
Magnitud
Surface
Pipeline Browser
builtin:
case0001.vtu
Object Inspector
Properties Display Information
Apply Reset Delete
Cell/Point Array Status
pressure
velocity
Selection Inspector
Current Object: case0001.vtu
Create Selection
Active Selection
Selected Object: [selected object name]
Selection Type: IDs
Field Type: CELL
Animation View
Mode: Sequence Time: 0 Start Time: 0 End Time: 1 No. Frames: 10
Time
TimeKeeper - Time
0.000e+00 2.500e-01 5.000e-01 7.500e-01 1.000e+00
case0001.vtu Cell Arrays

Exercise: Adding VTU output to the solver

- Choose an existing project
  - Perhaps not tutorial 3
  - Here Tutorial 4
- Add the definitions to your sif file either by ElmerGUI or manually
  - Remember that when you leave the GUI and edit the sif manually there is no going back
- Open the resulting .vtu file in Paraview and visualize the results