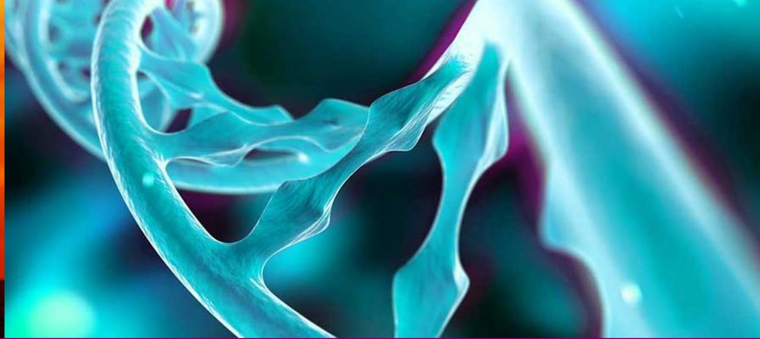


Elmer FEM course

Introduction to Elmer FEM software

May 23rd-24th, 2018

CSC – Suomalainen tutkimuksen, koulutuksen, kulttuurin ja julkishallinnon ICT-osaamiskeskus

CSC

facebook.com/CSCfi

twitter.com/CSCfi

youtube.com/CSCfi

linkedin.com/company/csc---it-center-for-science

github.com/CSCfi

Elmer FEM course

Introduction to Elmer FEM software

May 23rd-24th, 2018

Lecturers (in alphabetical order):

Peter Råback

Thomas Zwinger

About this course

Practicalities

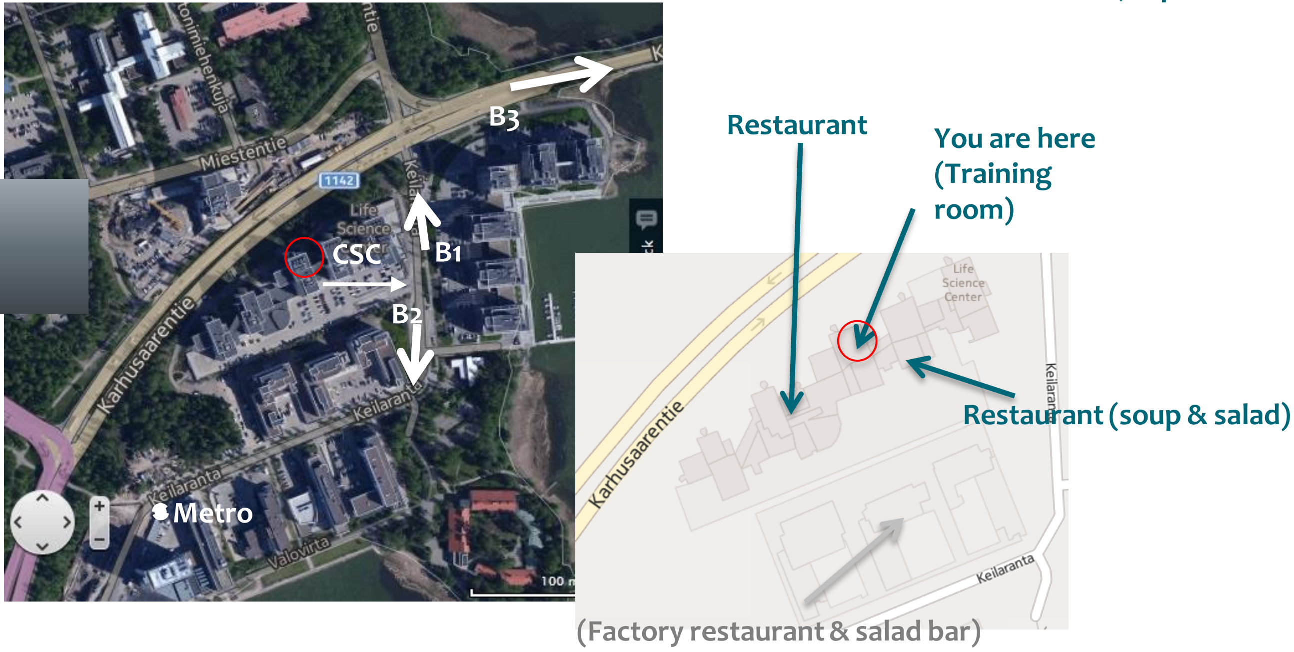
- Keep the name tag visible
- Lunch is served in the same building
- Toilets are in the lobby
- Network:
  - WIFI: eduroam, HAKA authentication or CSC-guest (with credentials in badge)
  - Ethernet cables on the tables
- Public transport:
  - Metro-> Helsinki/Center or Espoo
  - See next slide concerning bus-stops and lines

- **If you came by car**: parking is being monitored - ask for a temporary parking permit from the reception (tell which workshop you're participating)
- Visiting outside: doors by the reception desks are open, others are one-way (out)
- Room locked during lunch (save to leave computer, ...)
  - lobby open, use lockers
- Username and password for *workstations*: given on-site

Around CSC

B1 (555)→ Ottaniemi
B2 (555)→ Lauttasaari (also to Metro)
B3 (551) → Pasila
Metro → Helsinki, Espoo

Mind the changes introduced by the construction work



Program, May 23rd

09:00 – 09:30 Morning coffee & registration

09:30 – 09:45 Introduction to the course (whereabouts, etc.)

09:45 – 11:00 Introduction to Elmer finite element software

11:00 – 12:00 Hands-on session using ElmerGUI (hands-on)

12:15 – 13:15 Lunch

13:00 – 14:30 Elmer structure explained & hands-on session continued

14:30 – 15:00 Coffee

15:00 – 16:30 Alternative pre- and postprocessers

16:30 End day 1

Program, May 24th

09:00 – 10:00 The (Elmer)Solver Input File (SIF) explained including MATC (the poor-man's MatLab of Elmer)

10:00 – 10:30 Coffee

10:30 – 12:00 Programming user functions and solvers in Elmer

12:00 – 13:00 Lunch

13:00 – 13:45 Internal pre- and post-processing features in Elmer

13:45 – 14:30 Parallel computing with Elmer

14:30 – 15:00 Coffee

15:00 – 15:30 Parallel computing demo (if wanted – else we start with next point)

15:30 – 16:30 Miscellaneous topics & User problems (tell us all your Elmer problems)

What about you?

- Could we take a few minutes to get information on your ...
  - background
  - motivation to join that course
  - expectations to this course
  - anything else?

Introduction to Elmer FEM software

ElmerTeam
CSC – IT Center for Science, Finland

CSC, 2018

Non-profit state entity with special tasks

Turnover in year 2016

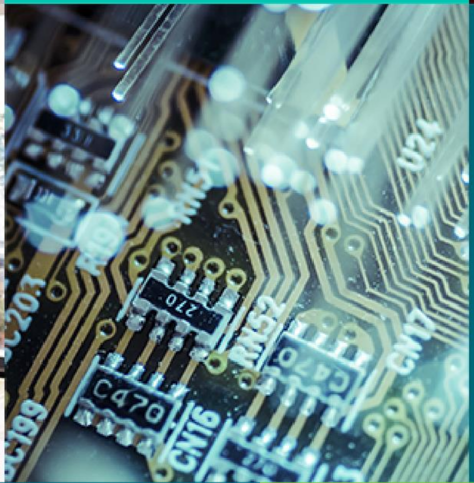
36.8M€

Office in Espoo, datacenter in Kajaani, Finland

Owned by state

(70%)

and all Finnish education higher institutions (30%)



Circa

300

employees

CSC's services

Computing and software

Data management and analytics for research

Support and training for research

Research administration

Solutions for managing and organizing education

Solutions for learners and teachers

Solutions for educational and teaching cooperation

Hosting services tailored to customers' needs

Identity and authorisation

Management and use of data

ICT platforms, Funet network and data center functions are the base for our solutions

Funet – National and International Networks and Services

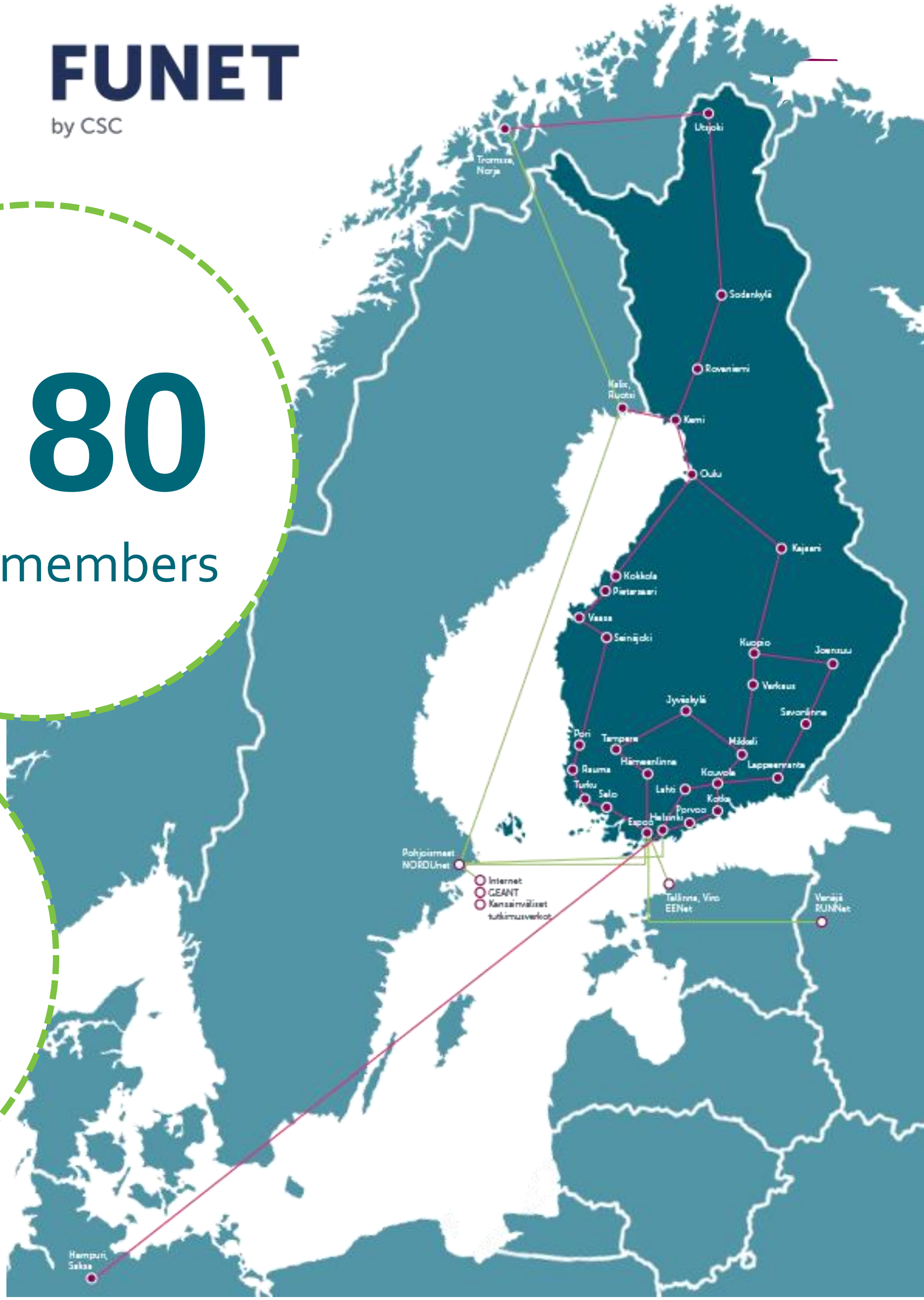
FUNET
by CSC

Services included in Funet membership

- Funet Network Connections
- Funet CERT Information Security Service
- Vulnerability Scanner
- Certificate Service
- eduroam Roaming Access Service
- Funet FileSender File Sharing Service

Services with additional costs

- Funet Etuubi Video Management System
- Funet Silta Video Conferencing MCU Service
- Funet Tiimi Web Conferencing System
- Funet light Paths
- Router Service
- Streaming Service



ca. 80
Funet members

370 000
end-users

CSC datacenter in Kajaani

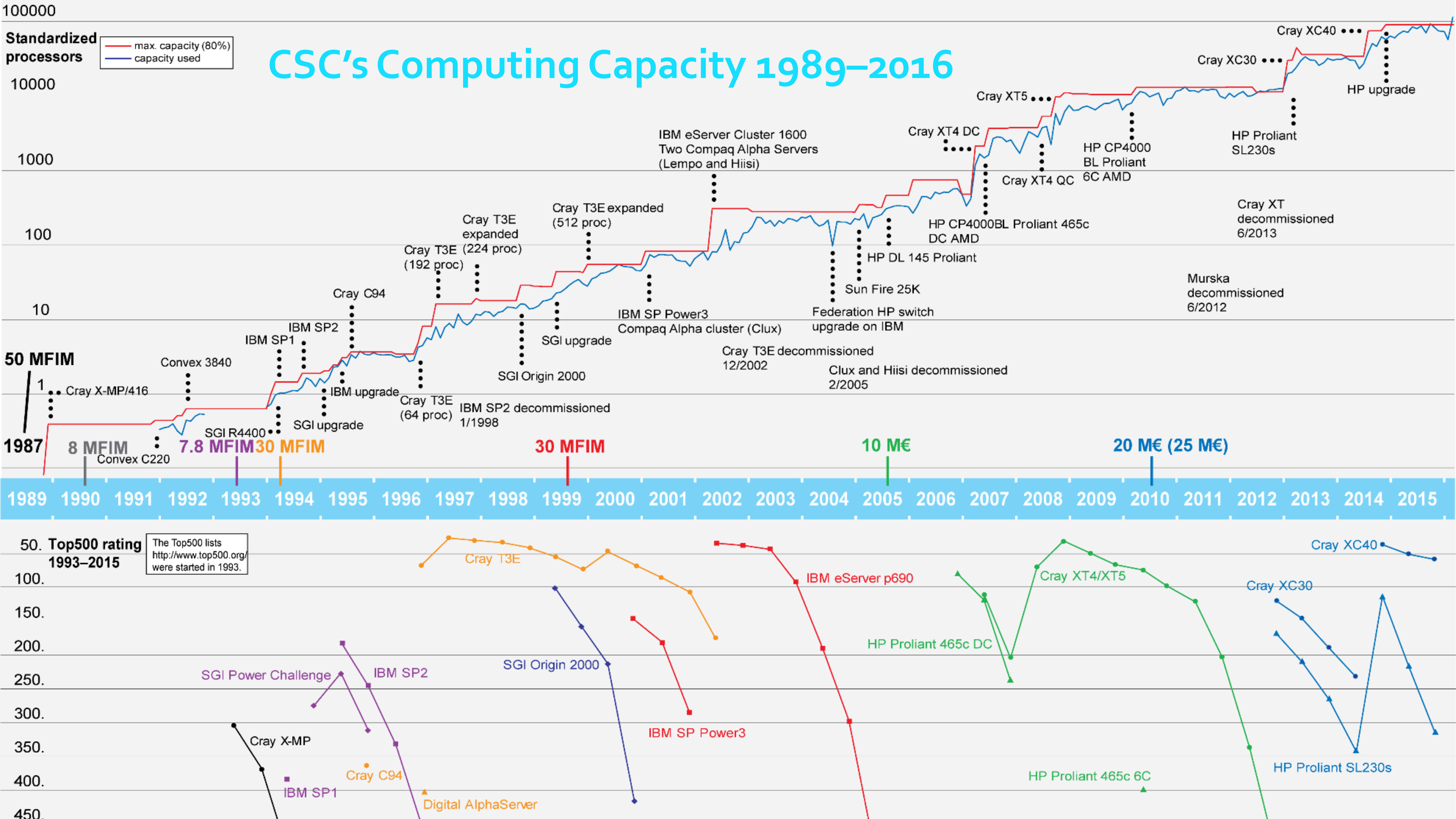
- 3 000 m2 (option to 4 000 m2 additional datacenter space)
- Redundant green power scalability up to hundreds of MW, based on customer need. Existing power capacity: 10 MW (redundant)
- Local and competent partner network guarantees rapid scalability and secure operations
- State-of-the-art datacenter technology (modularity of the datacenter, easy expansion, free air cooling all year round) delivering world-class eco-efficiency and zero carbon footprint. Annual PUE 1,04 (2016)
- Also traditional water cooling datacenter facilities are available for certain supercomputer type of services
- High-end availability on both power supply, cooling and core network connectivity

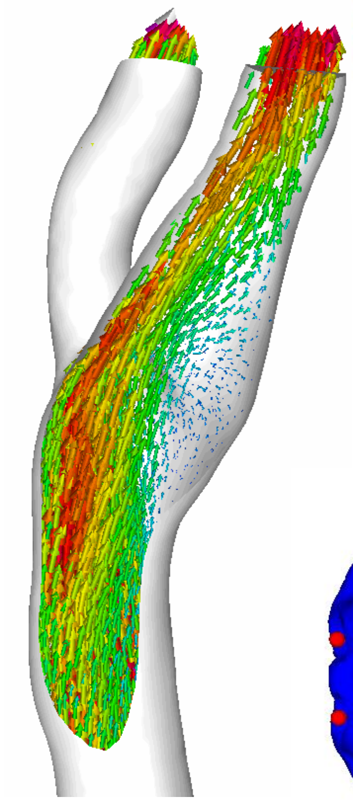
CSC's Computing Services

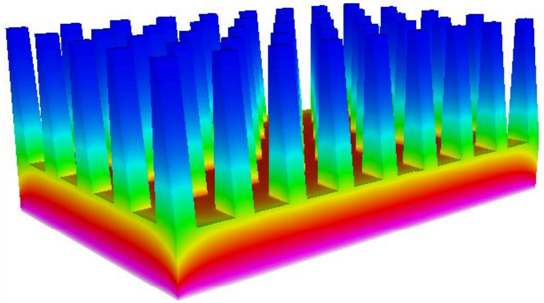
Performance	Capacity	Special processors	Cloud	Hosting
Sisu (Cray) 40 512 cores Aries interconnect	Taito (HP) 18 816 cores InfiniBand interconnect	Taito extension (Bull) 996 cores 76 Nvidia K40 GPUs 90 Intel Xeon Phi 7120X	cPouta Cores provisioned from Taito cluster	Kajaani Espoo

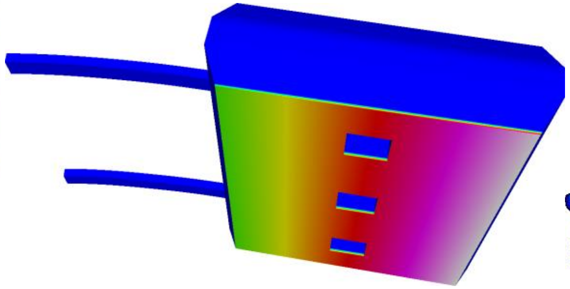
>4PB, ~ 100GB/s

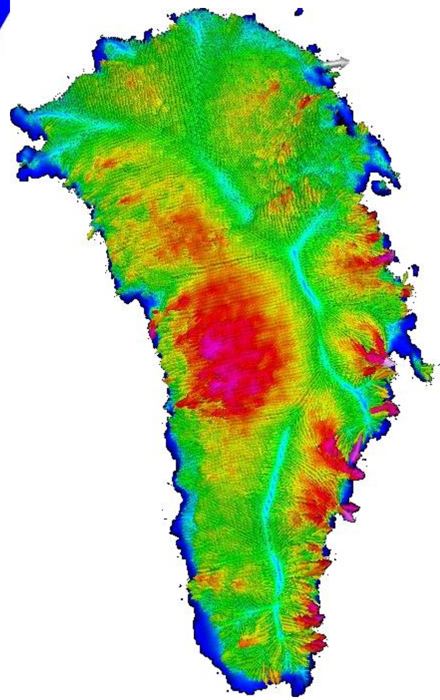
Storage services

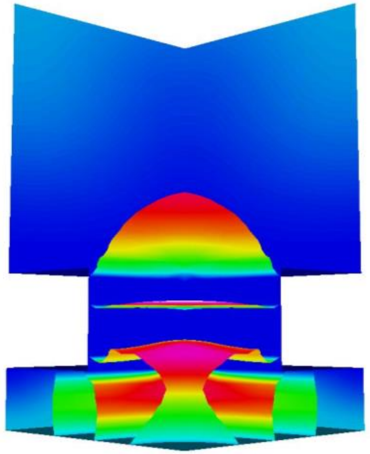

CSC's Computing Capacity 1989–2016
Standardized processors
max. capacity (80%)
capacity used
100000
10000
1000
100
10
1
1987
50 MFIM
8 MFIM
7.8 MFIM
30 MFIM
30 MFIM
10 M€
20 M€ (25 M€)
Cray X-MP/416
Convex C220
Convex 3840
SGI R4400
IBM SP1
IBM SP2
IBM upgrade
SGI upgrade
Cray C94
Cray T3E (64 proc)
Cray T3E (192 proc)
Cray T3E expanded (224 proc)
SGI Origin 2000
SGI upgrade
Cray T3E expanded (512 proc)
IBM SP2 decommissioned 1/1998
IBM SP Power3
Compaq Alpha cluster (Clux)
IBM eServer Cluster 1600
Two Compaq Alpha Servers
(Lempo and Hiisi)
Cray T3E decommissioned 12/2002
Federation HP switch upgrade on IBM
Clux and Hiisi decommissioned 2/2005
Sun Fire 25K
HP DL 145 Proliant
HP CP4000BL Proliant 465c DC AMD
Cray XT4 DC
Cray XT4 QC
Cray XT5
HP CP4000 BL Proliant 6C AMD
Murska decommissioned 6/2012
Cray XT decommissioned 6/2013
HP Proliant SL230s
Cray XC30
HP upgrade
Cray XC40
1989 1990 1991 1992 1993 1994 1995 1996 1997 1998 1999 2000 2001 2002 2003 2004 2005 2006 2007 2008 2009 2010 2011 2012 2013 2014 2015
Top500 rating 1993–2015
The Top500 lists http://www.top500.org/ were started in 1993.
50.
100.
150.
200.
250.
300.
350.
400.
450.
Cray X-MP
IBM SP1
SGI Power Challenge
IBM SP2
Cray C94
Digital AlphaServer
Cray T3E
SGI Origin 2000
IBM SP Power3
IBM eServer p690
HP Proliant 465c DC
Cray XT4/XT5
HP Proliant 465c 6C
Cray XC30
HP Proliant SL230s
Cray XC40



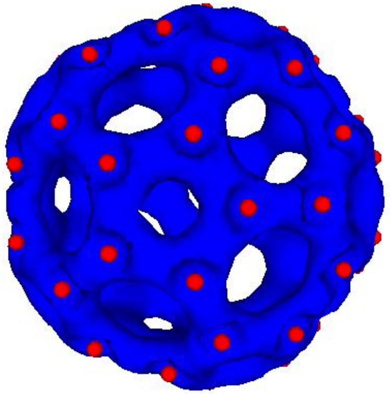


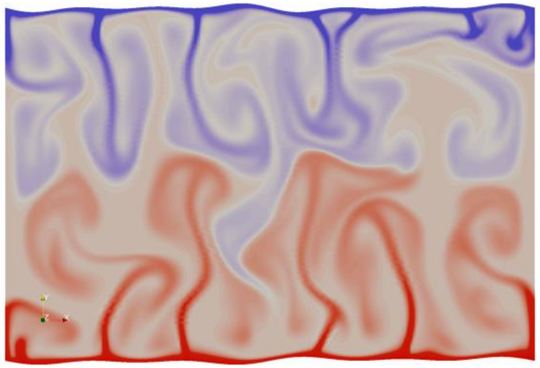


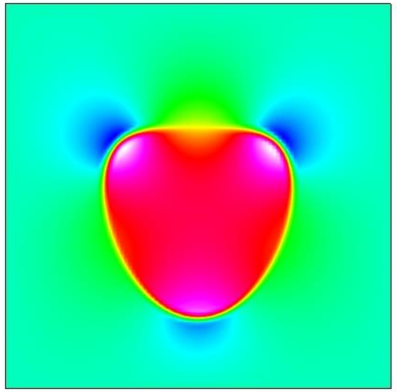


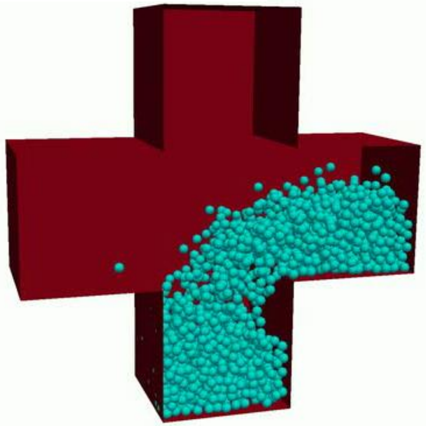


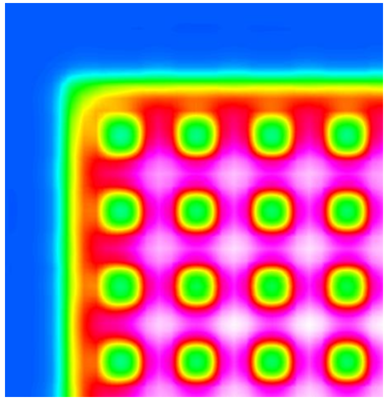
Elmer finite element software for multiphysical problems

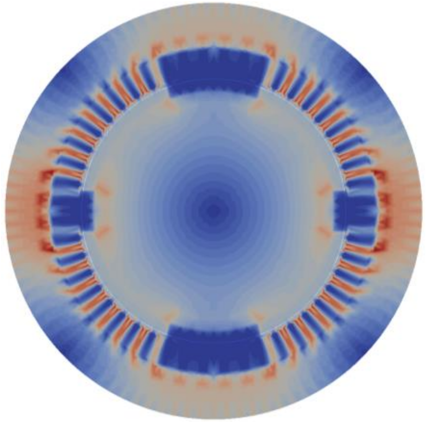


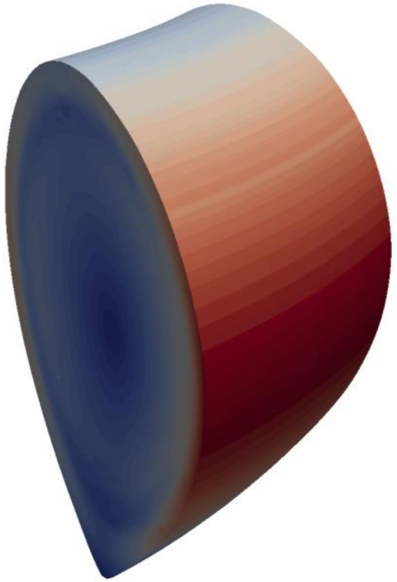








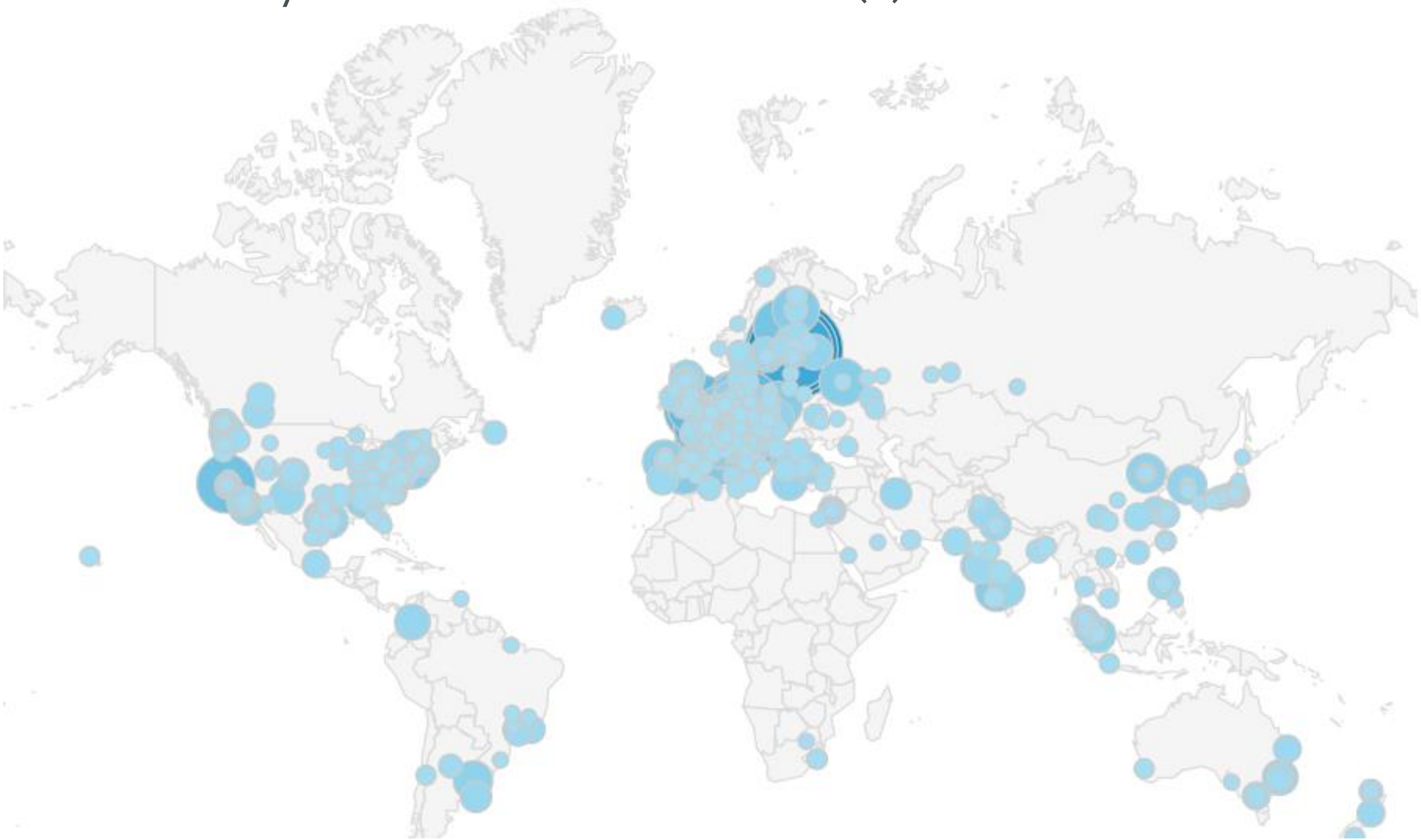




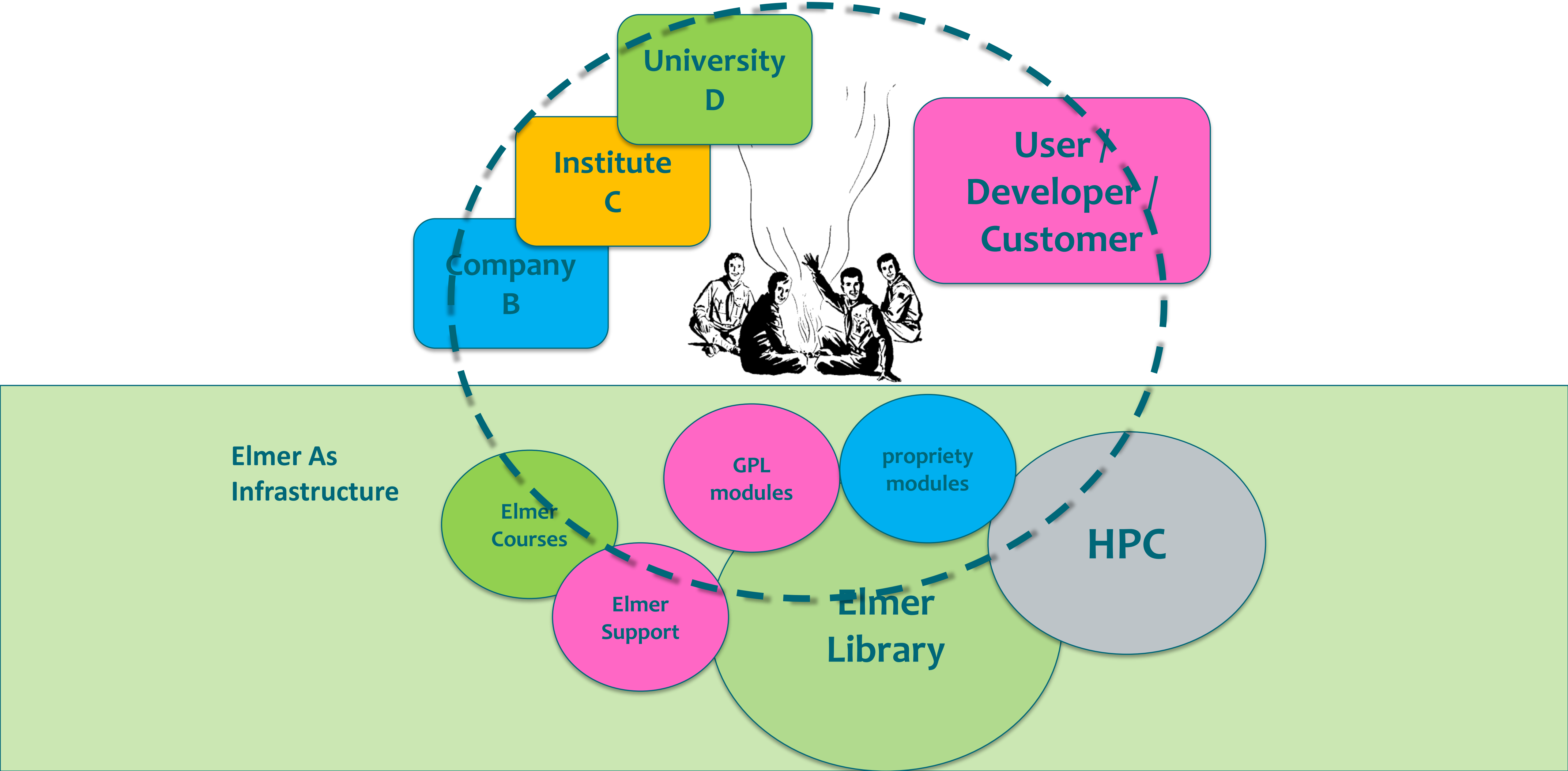
Figures by Esko Järvinen, Mikko Lyly, Peter Råback, Timo Veijola (TKK) & Thomas Zwinger

Elmer was published under GPL on sf.net in 2008

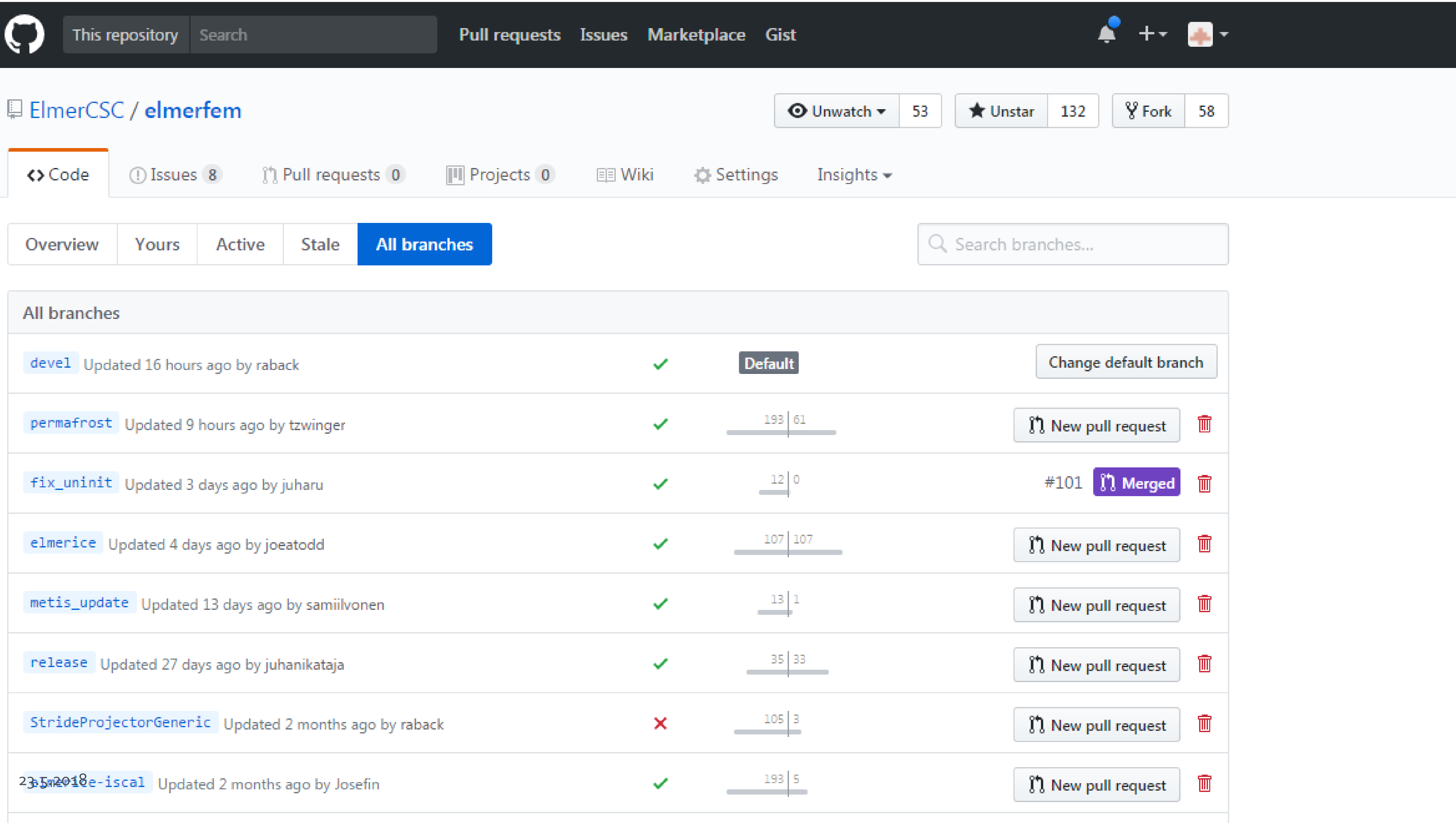
- Used worldwide by thousands of researchers (?)



Elmer – Infrastructure for Open Research



Elmer is hosted at GitHub and accepts contributions



Elmer in numbers

Software

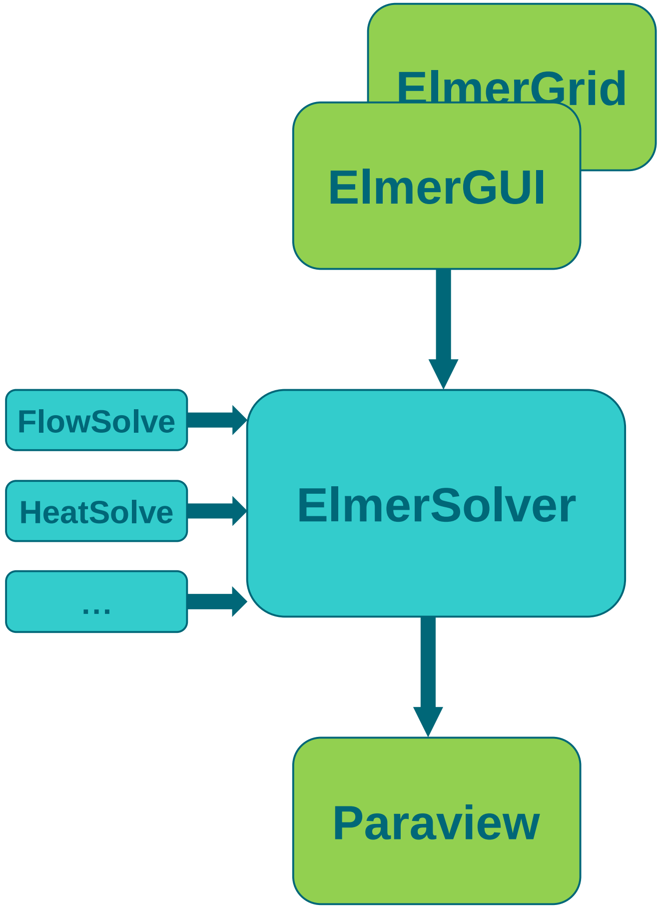
- ~400,000 lines of active code
  - ~3/4 in Fortran, 1/4 in C/C++
- ~580 consistency tests
- ~750 pages of documentation
- ~1000 code commits yearly

Community

- ~20,000 downloads for Windows binary yearly
  - Linux users untracked
- ~2000 forum postings yearly
- ~100 people participate on Elmer courses yearly
- Several Elmer related scientifc visits to CSC yearly

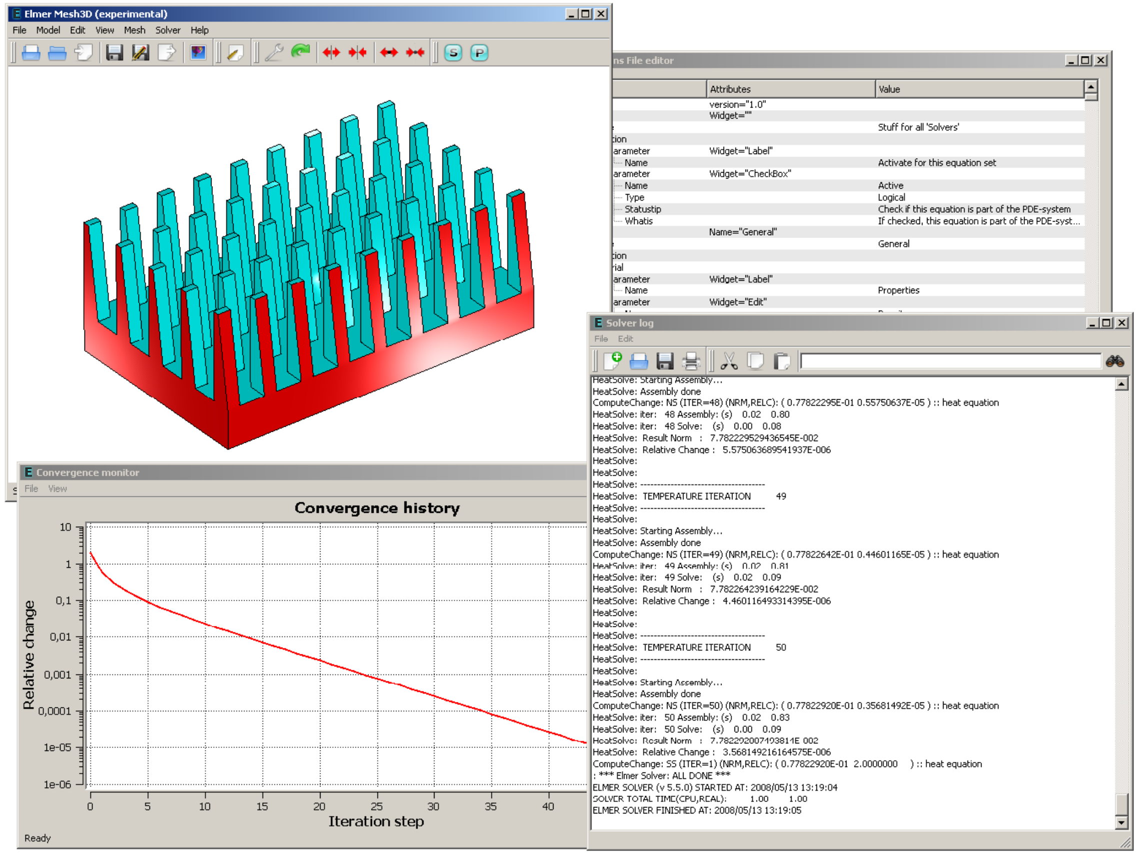
Elmer finite element software

- **Elmer** is actually a suite of several programs
  - Components may also be used independently
- **ElmerGUI** – Preprocessing
- **ElmerSolver** – FEM Solution
  - Each physical equation is a **dynamically loaded** library to the main program
- **ElmerGrid** - structured meshing, mesh import & partitioning
- **ElmerPost** - Postprocessing
  - made obsolite by Paraview



ElmerGUI

- Graphical user interface of Elmer
  - Based on the **Qt** library (GPL)
  - Developed at CSC since 2/2008
- Mesh generation
  - Plugins for Tetgen, Netgen, and ElmerGrid
  - CAD interface based on OpenCascade
- Easiest tool for case specification
  - Even educational use
  - Parallel computation
- New solvers easily supported through GUI
  - XML based menu definition



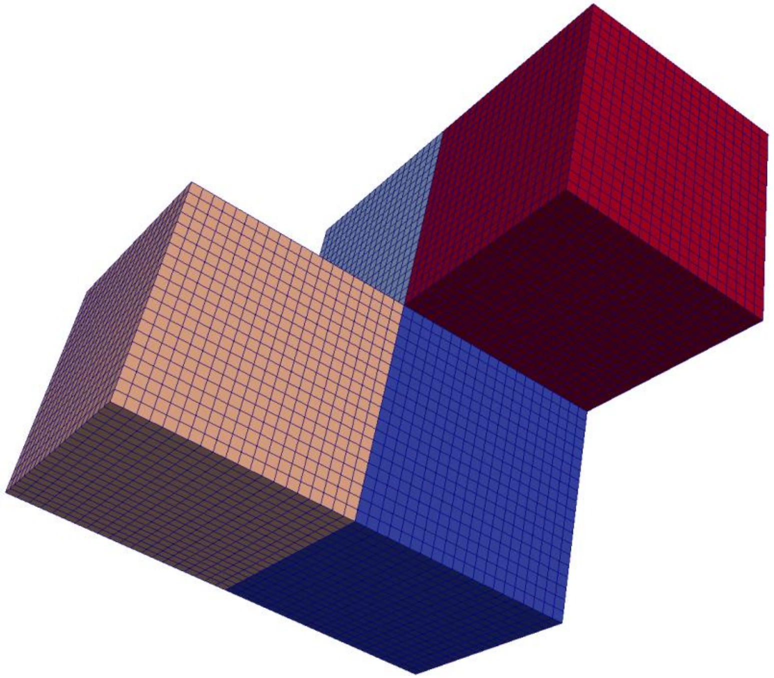
ElmerSolver

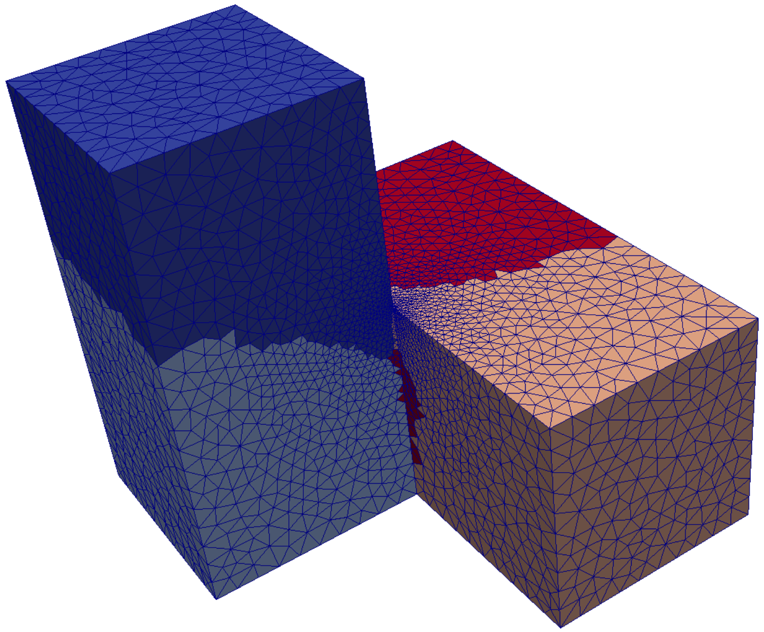
- Assembly and solution of the finite element equations and beyond
- Large number of auxiliary routines
- Note: When we talk of Elmer we mainly mean ElmerSolver
- ~95% of development effort

```
ELMER SOLVER (v 8.3) STARTED AT: 2017/06/19 18:35:01
ParCommInit:  Initialize #PEs:          1
MAIN: ==============================================================
MAIN: ElmerSolver finite element software, Welcome!
MAIN: This program is free software licensed under (L)GPL
MAIN: Copyright 1st April 1995 - , CSC - IT Center for Science Ltd.
MAIN: Webpage http://www.csc.fi/elmer, Email elmeradm@csc.fi
MAIN: Version: 8.3 (Rev: 8068c86, Compiled: 2017-06-18)
MAIN:  HYPRE library linked in.
MAIN:  MUMPS library linked in.
MAIN: ==============================================================
MAIN: Reading Model: flux.sif
LoadMesh: Base mesh name: ./angle
MAIN: -------------------------------------
Loading user function library: [HeatSolve]...[HeatSolver]
HeatSolve: -------------------------------------
HeatSolve:  TEMPERATURE ITERATION         1
HeatSolve: -------------------------------------
HeatSolve: Assembly:
DefUtils::DefaultDirichletBCs: Setting Dirichlet boundary conditions
ComputeChange: NS (ITER=1) (NRM,RELC): ( 0.25941344E-01  2.0000000     ) :: heat equation
CompareToReferenceSolution: Solver 1 PASSED:  Norm = 2.59413436E-02  RefNorm = 2.5941343
CompareToReferenceSolution: Relative Error to reference norm: 1.512027E-09
CompareToReferenceSolution: PASSED all 1 tests!
ElmerSolver: *** Elmer Solver: ALL DONE ***
ElmerSolver: The end
SOLVER TOTAL TIME(CPU,REAL):        0.10       0.15
ELMER SOLVER FINISHED AT: 2017/06/20 01:35:01
```

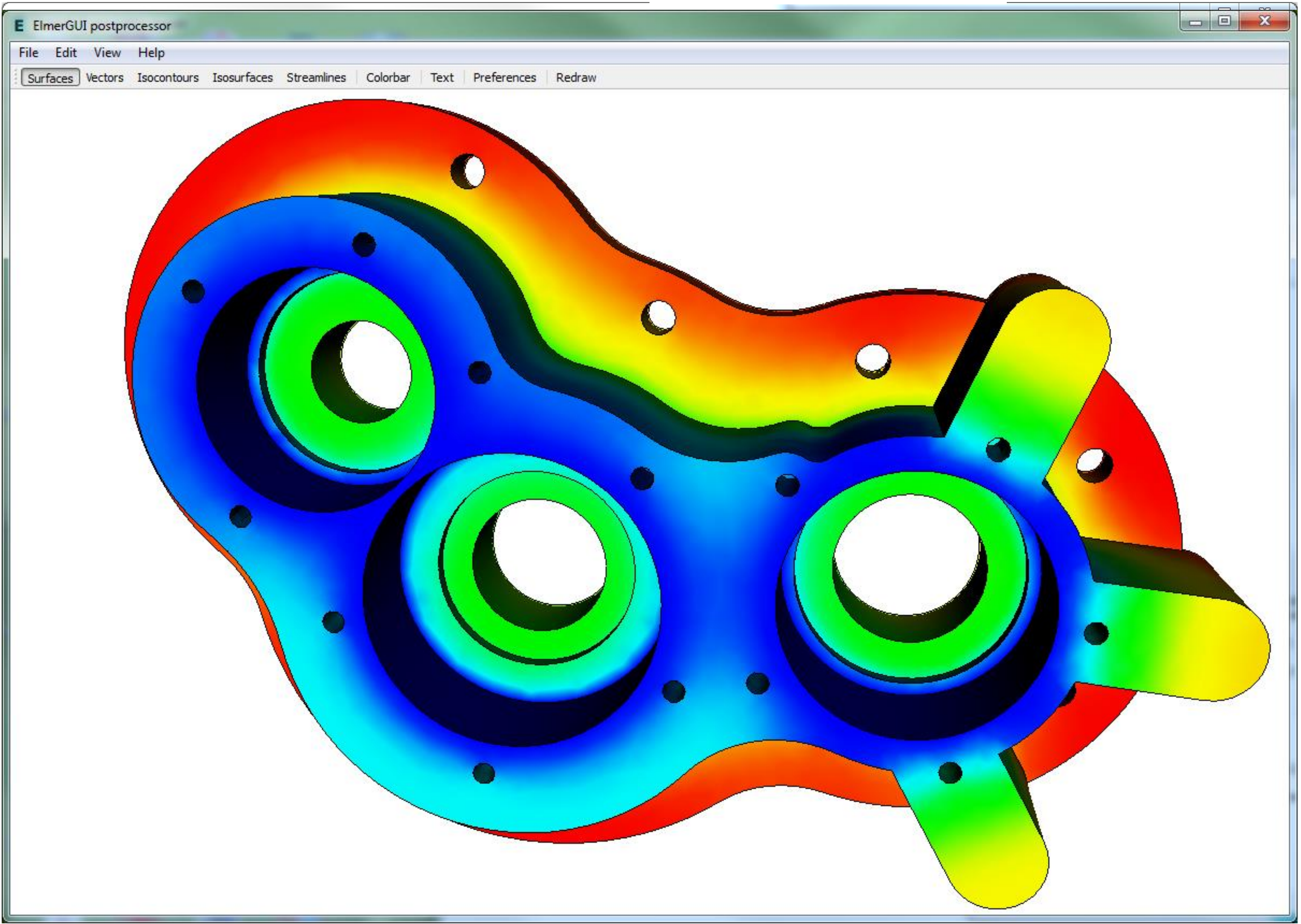
ElmerGrid (standalone + built-in ElmerGUI)

- Creation of 2D and 3D structured meshes
  - Rectangular basic topology + simple mapping
  - Extrusion, rotation
- Mesh Import
  - About ten different formats:
    Ansys, Abaqus, Fidap, Comsol, Gmsh,...
- Mesh manipulation
  - Increase/decrease order
  - Scale, rotate, translate
- Partitioning
  - Simple geometric (upper figure)
  - Metis library (lower figure)





SERIAL WORKFLOW: VISUALIZATION



$$\Omega = \bigcup \Omega_e$$

$$-\nabla \cdot \kappa \nabla T = h \ \text{ in } \Omega$$

$$T = T_0 \ \text{ at } \Gamma$$

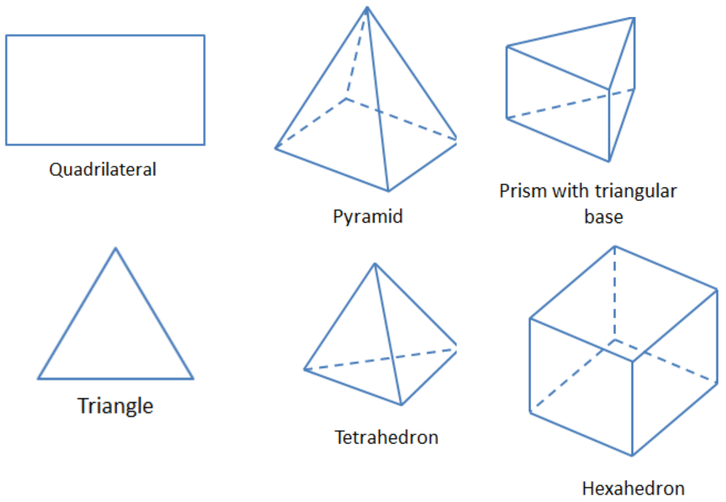
$$A_{ij} += \int \kappa \nabla \varphi_i \cdot \nabla \varphi_j \, d\Omega_e$$

$$b_i += \int h \varphi_i \, d\Omega_e$$

$$x = A^{-1} b$$

ElmerSolver – Numerical Methods

- Time-dependency
  - Static, transient, harmonic, eigenmode, scanning
- Discretization
  - Element families: nodal, edge (Hcurl), face (Hdiv), and p-elements, DG
  - Element shapes: triangles, quads, tets, wedges, pyramids, hexas
  - Formulations: Galerkin, stabilization, bubbles
  - Continuity: Mortar finite elements for periodic and nonconforming meshes



- Linear system solvers
  - Direct: Lapack, Umfpack, (SuperLU, Mumps, Pardiso)
  - Iterative Krylov space methods (HutIter & Hypre)
  - multigrid solvers (GMG & AMG) for “easy” equations (own & Hypre)
  - Preconditioners: ILU, BILU, Parasails, multigrid, SGS, Jacobi,…
- Adaptivity
  - For selected equations, unfortunately no parallel implementation

ElmerSolver - Physical Models

- Heat transfer
  - ✓ Heat equation
  - ✓ Radiation with view factors
  - ✓ convection and phase change
- Fluid mechanics
  - ✓ Navier-Stokes (2D & 3D)
  - ✓ RANS: *SST k-Ω, k-ε, v^2-f*
  - ✓ LES: VMS
  - ✓ Thin films: Reynolds (1D & 2D)
- Structural mechanics
  - ✓ General elasticity (unisotropic, lin & nonlin)
  - ✓ Plates & Shells
- Acoustics
  - ✓ Helmholtz
  - ✓ Linearized time-harmonic N-S
  - ✓ Monolithic thermal N-S
- Species transport
  - ✓ Generic convection-diffusion equation
- Electromagnetics
  - ✓ Solvers for either scalar or vector potential (nodal elements)
  - ✓ Edge element based AV solver for magnetic and electric fields
- Mesh movement (Lagrangian)
  - ✓ Extending displacements in free surface problems
  - ✓ ALE formulation
- Level set method (Eulerian)
  - ✓ Free surface defined by a function
- Electrokinetics
  - ✓ Poisson-Boltzmann
- Thermoelectricity
- Quantum mechanics
  - ✓ DFT (Kohn Scham)
- Particle Tracker

Poll on application fields (status 4/2018)

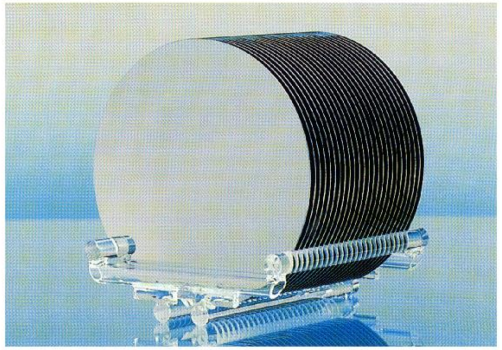
CSC

What are your main application fields of Elmer?

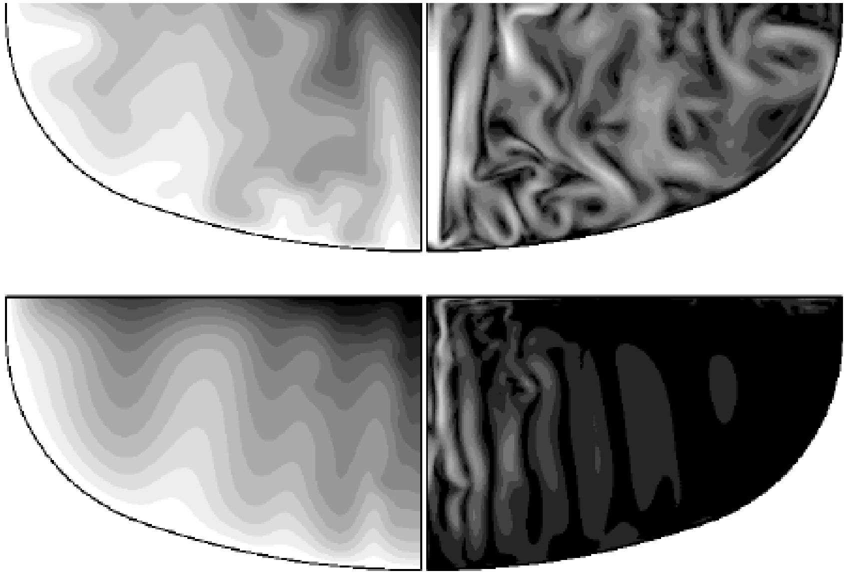
Heat transfer	70	28%
Fluid mechanics	65	26%
Solid mechanics	50	20%
Electromagnetics	45	18%
Quantum mechanics	5	2%
Something else (please specify)	14	6%
	Total votes : 249	

Czockralski Crystal Growth

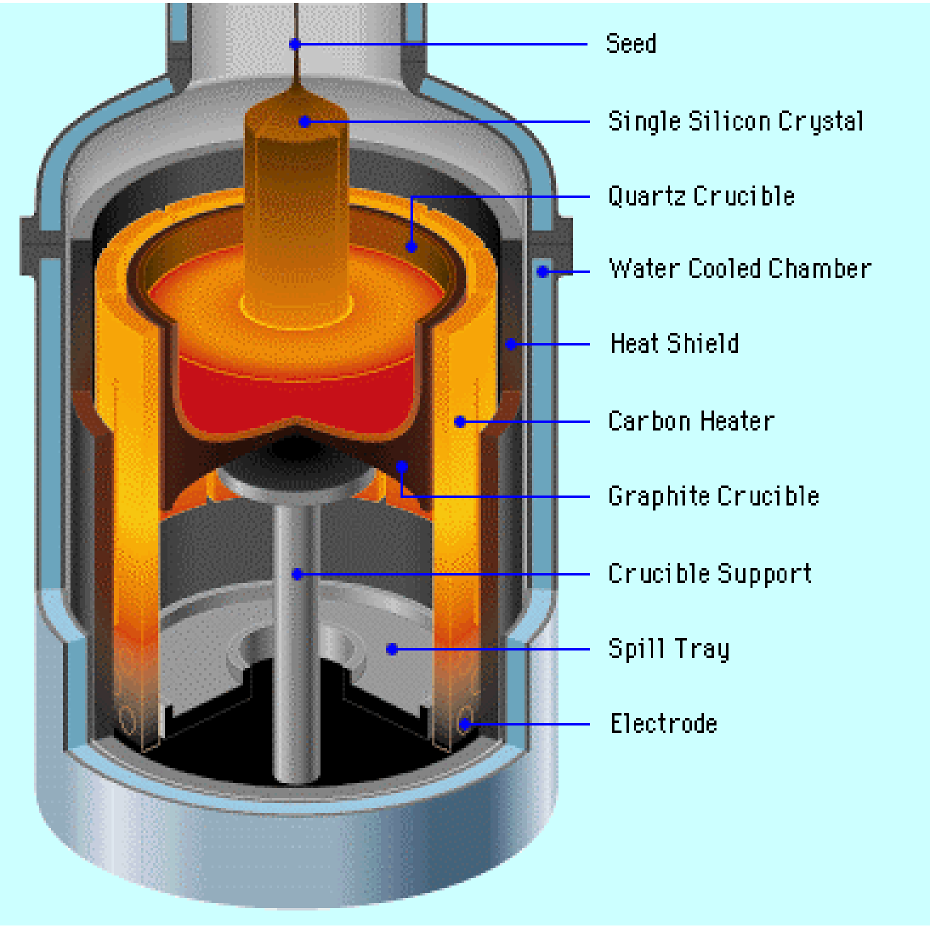
- Most crystalline silicon is grown by the Czhockralski (CZ) method
- One of the key application when Elmer development was started.



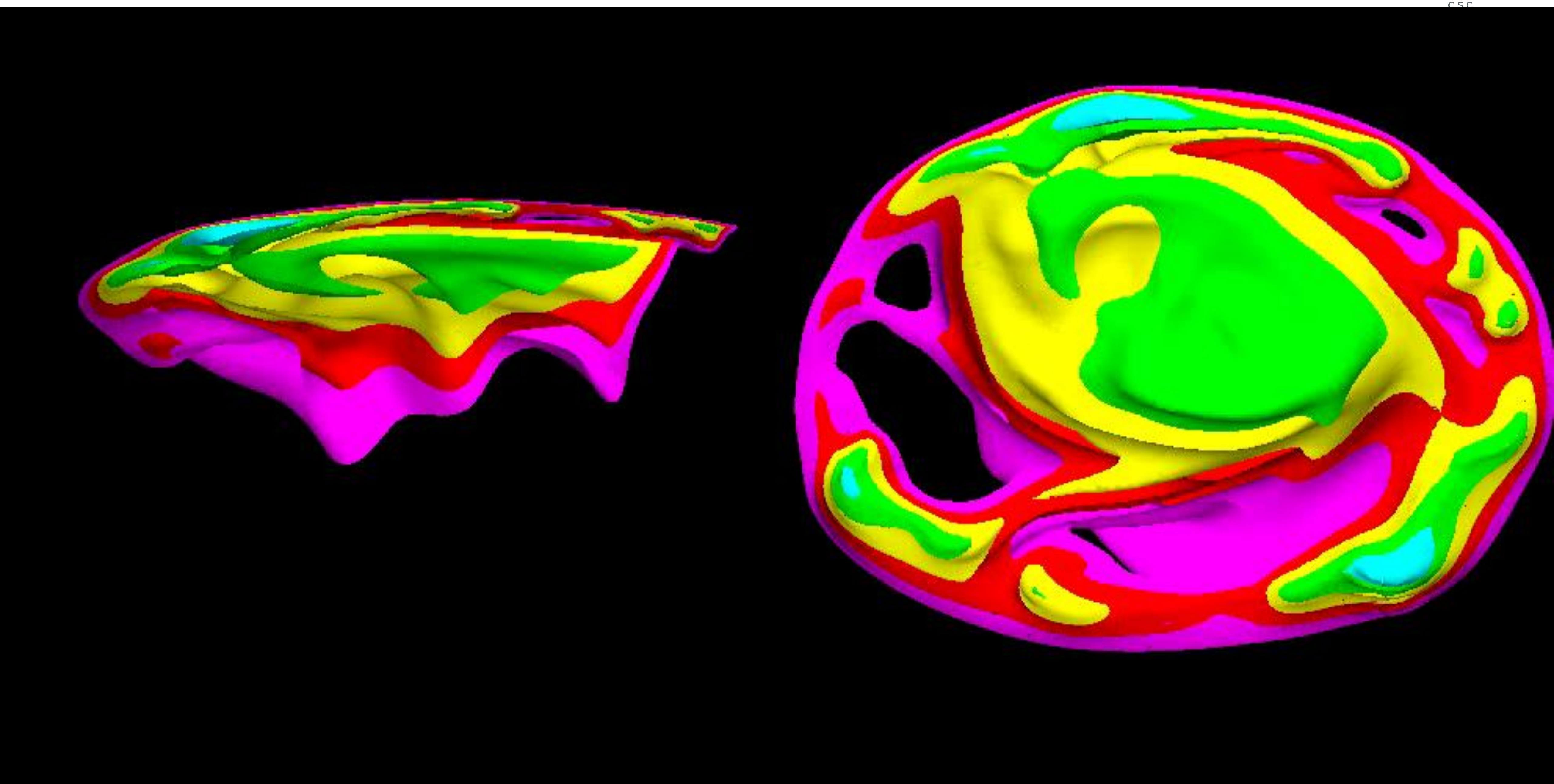
Figures by Okmetic Ltd.



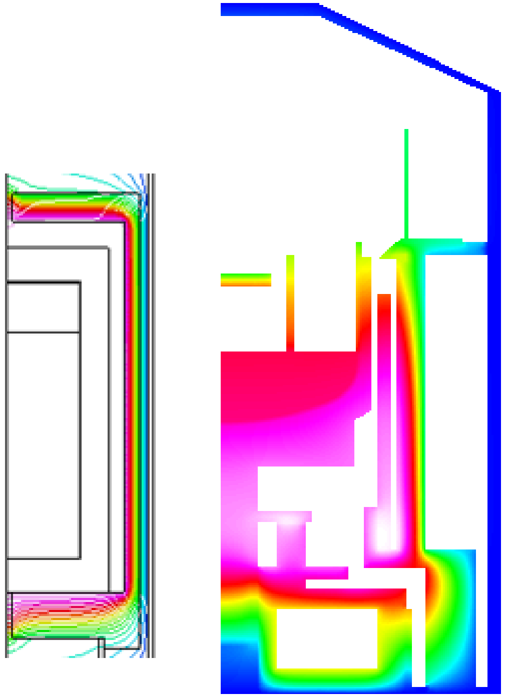
V. Savolainen et al., *Simulation of large-scale silicon melt flow in magnetic Czochralski growth,* J. Crystal Growth 243 (2002), 243-260.

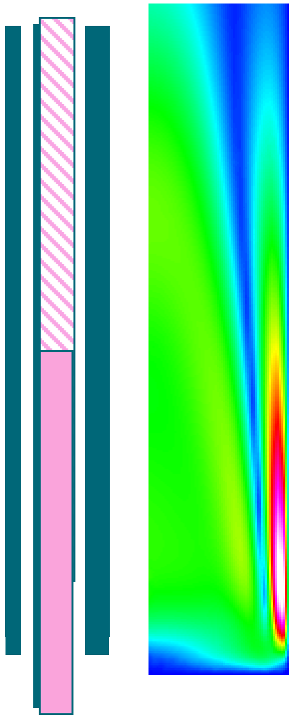


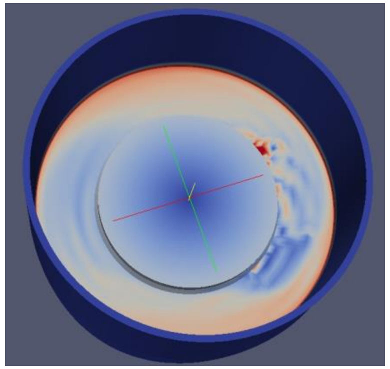
CZ-growth: Transient simulation

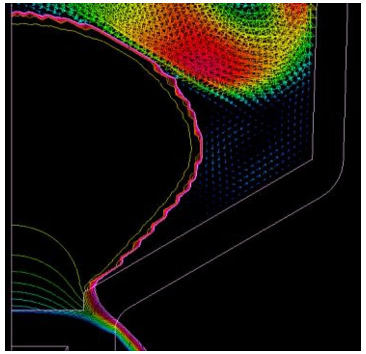


Elmer in Crystal Growth Simulations









- Elmer has been used extensively in crystal growth simulations: These include crystal and tube growth for silicon, silicon-carbide, NiMnGa and sapphire in Czochralski, HTCVD, sublimation, Bridgman, Vertical Gradient Freeze and Heat Exchanger Methods.
- Numerical results have been successfully verified with experiments.
- Elmer is a part of open-source chain from CAD to visualization, and offers an access to parallelism and a number of simultaneous simulations important for industrial R&D.

Silicom

Simulations Jari Järvinen, Silicom Oy, 2014

MEMS: Inertial sensor

- MEMS provides an ideal field for multi-physical simulation software
- Electrostatics, elasticity and fluid flow are often inherently coupled
- Example shows the effect of holes in the motion of an accelerometer prototype

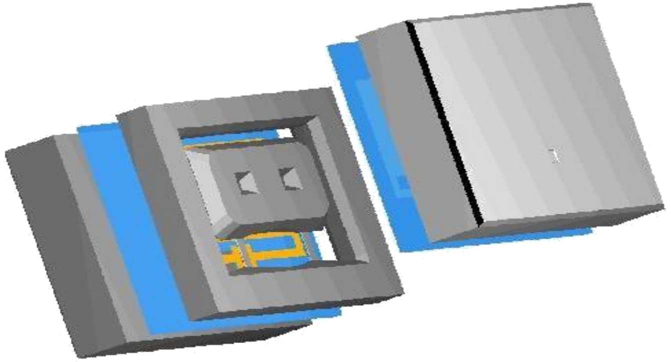
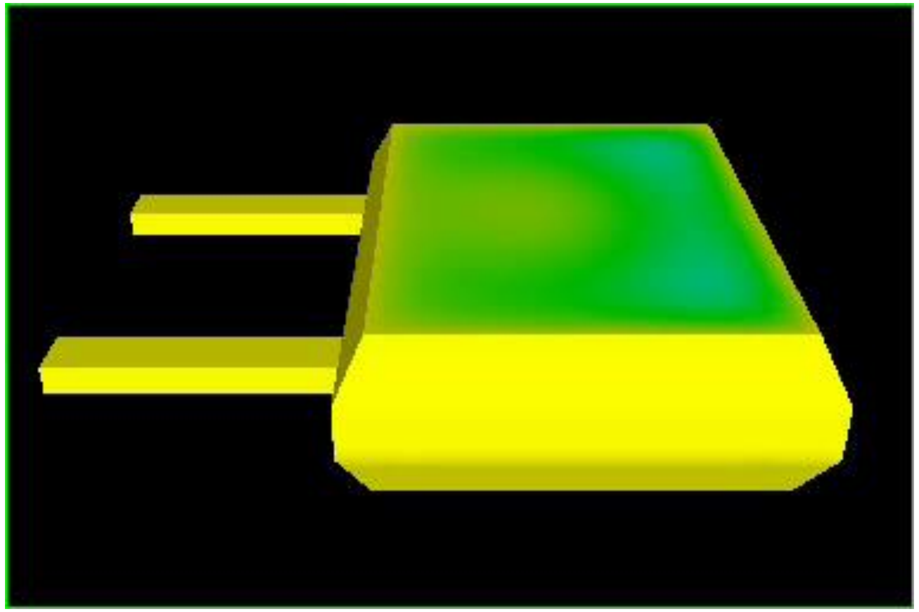
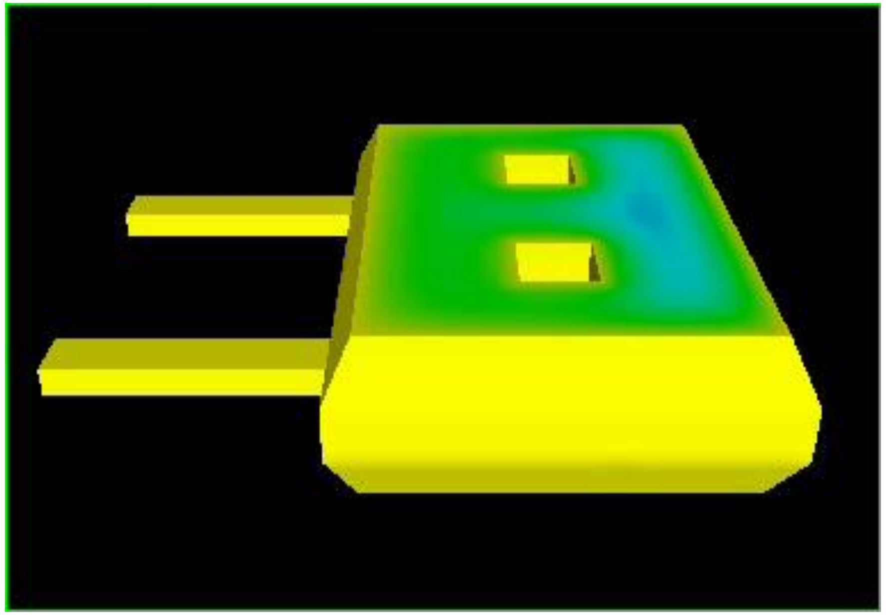


Figure by VTI Technologies

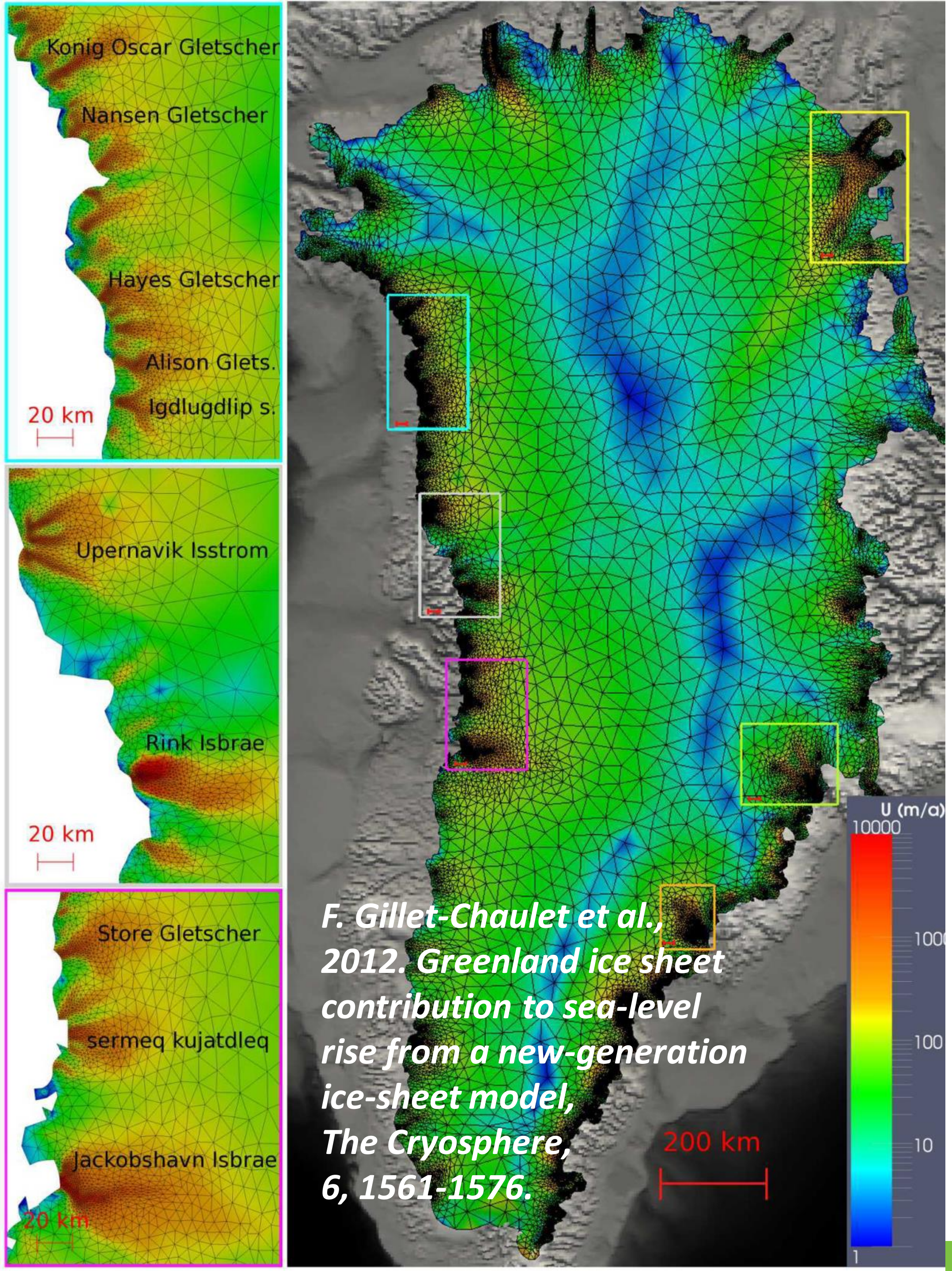




A. Pursula, P. Råback, S. Lähteenmäki and J. Lahdenperä, *Coupled FEM simulations of accelerometers including nonlinear gas damping with comparison to measurements*, J. Micromech. Microeng. **16** (2006), 2345-2354.

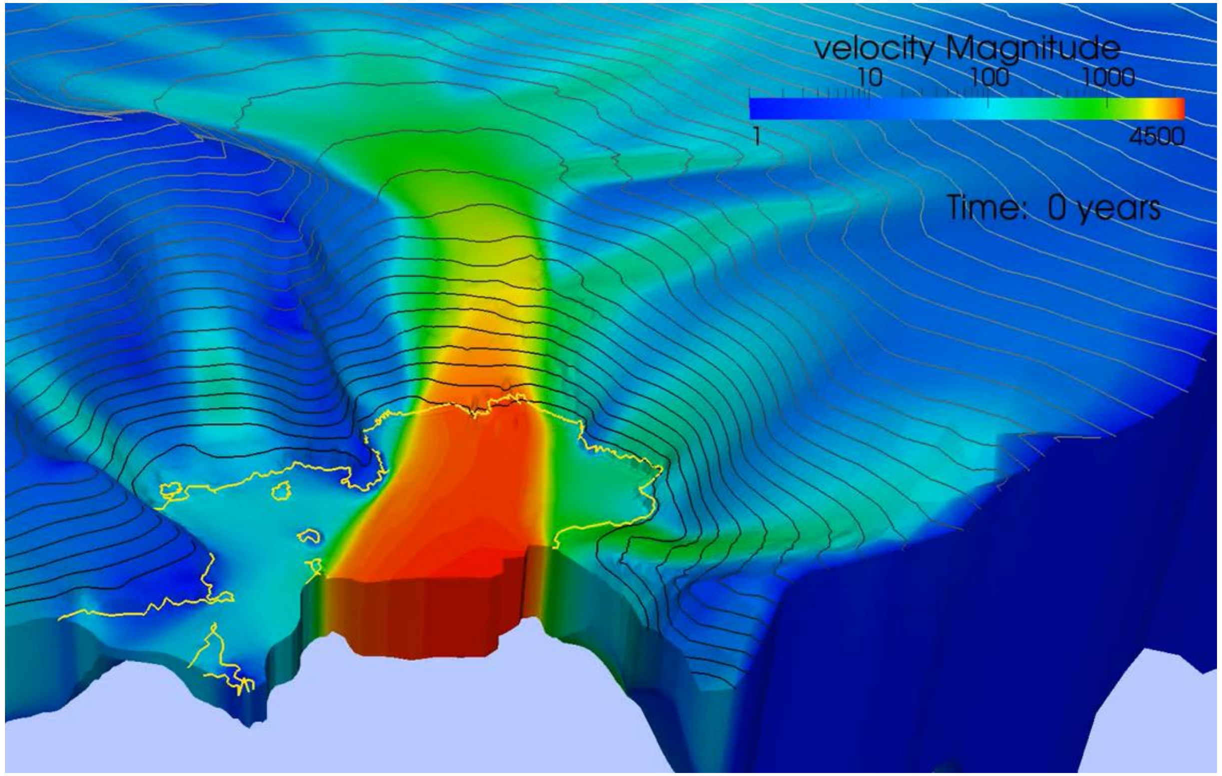
Elmer/ICE: Glaceology

- **Elmer/Ice** is the leading software used in 3D computational glaciology
- Full 3D Stokes equation to model the flow
- Large number of tailored models to deal with the special problems
- Motivated by climate change and sea level rise
- Currently ~100 peer-reviewed publications in the area
- Dedicated community portal `elmerice.elmerfem.org`



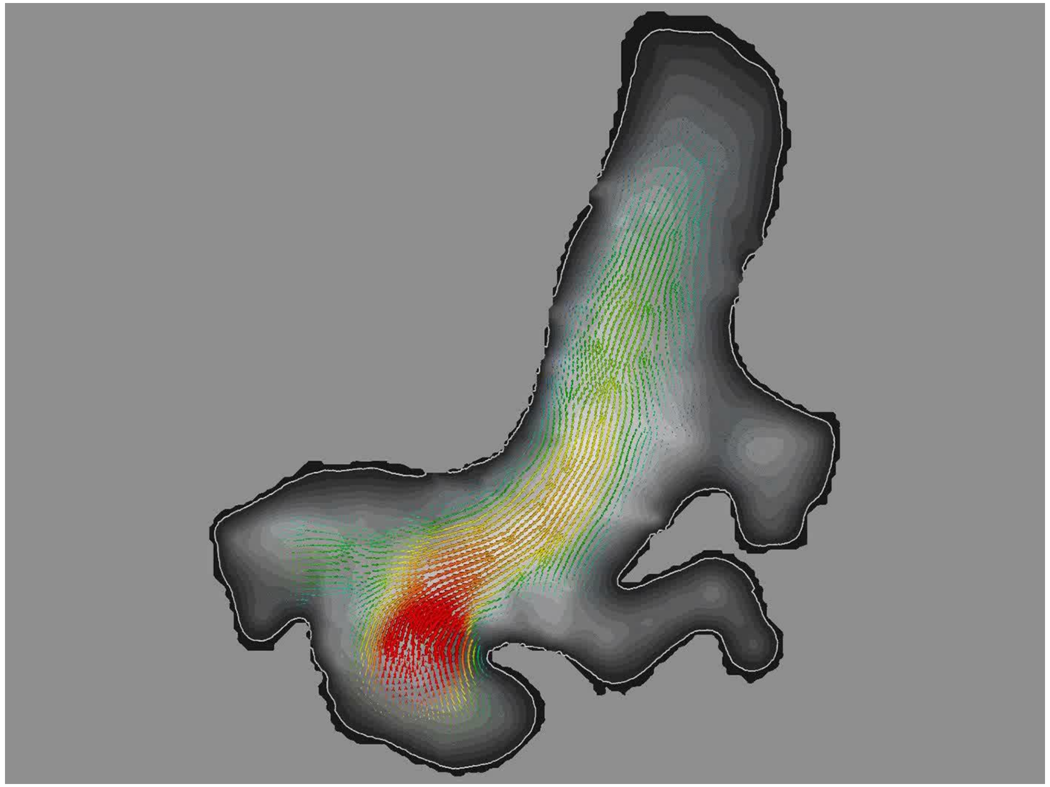
F. Gillet-Chaulet et al., 2012. Greenland ice sheet contribution to sea-level rise from a new-generation ice-sheet model, The Cryosphere, 6, 1561-1576.

Marine Ice Sheets



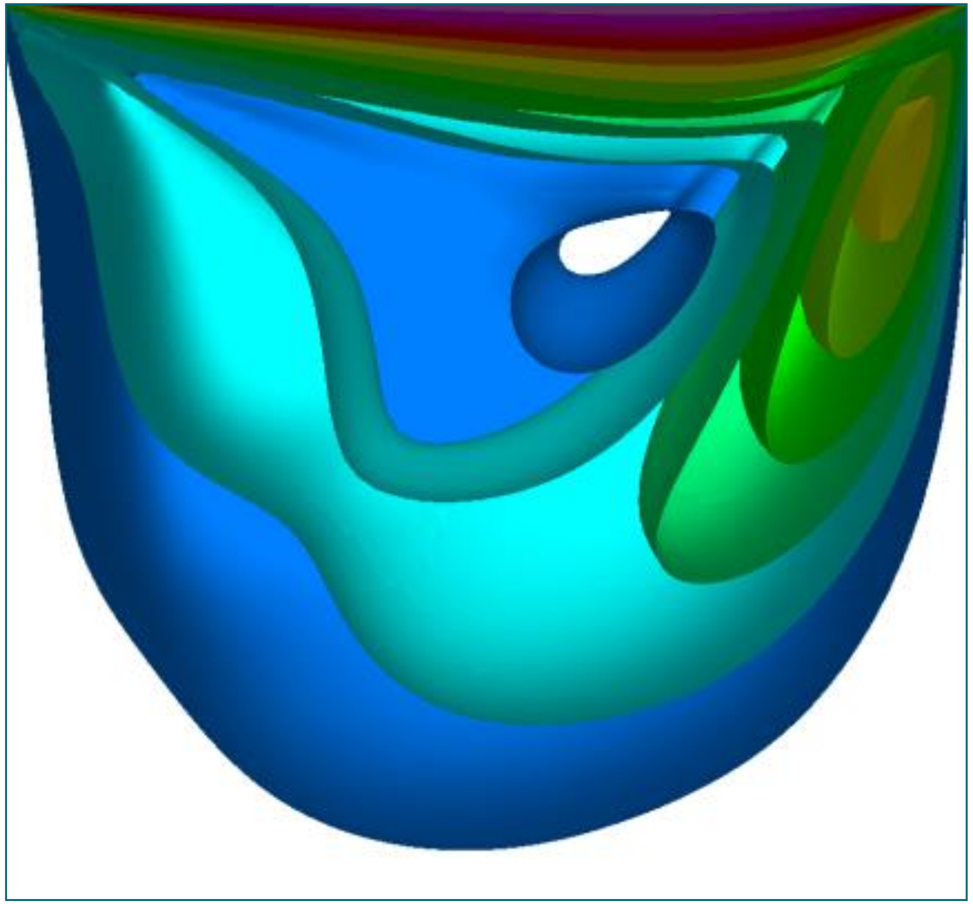
Favier, L., G. Durand, S. L. Cornford, G. H. Gudmundsson, O. Gagliardini, F. Giller-Chaulet, T. Zwinger, A. J. Payne and A. M. Le Brocq, 2014. ***Retreat of Pine Island Glacier controlled by marine ice-sheet instability***, Nature Climate Change

Glaciers



T. Zwinger and Moore, J. C. (2009) ***Diagnostic and prognostic simulations with a full Stokes model accounting for superimposed ice of Midtre Lovénbreen****, Svalbard*, The Cryosphere, 3, 217-229, doi:10.5194/tc-3-217-2009

Block preconditioning: Weak scaling of 3D driven-cavity



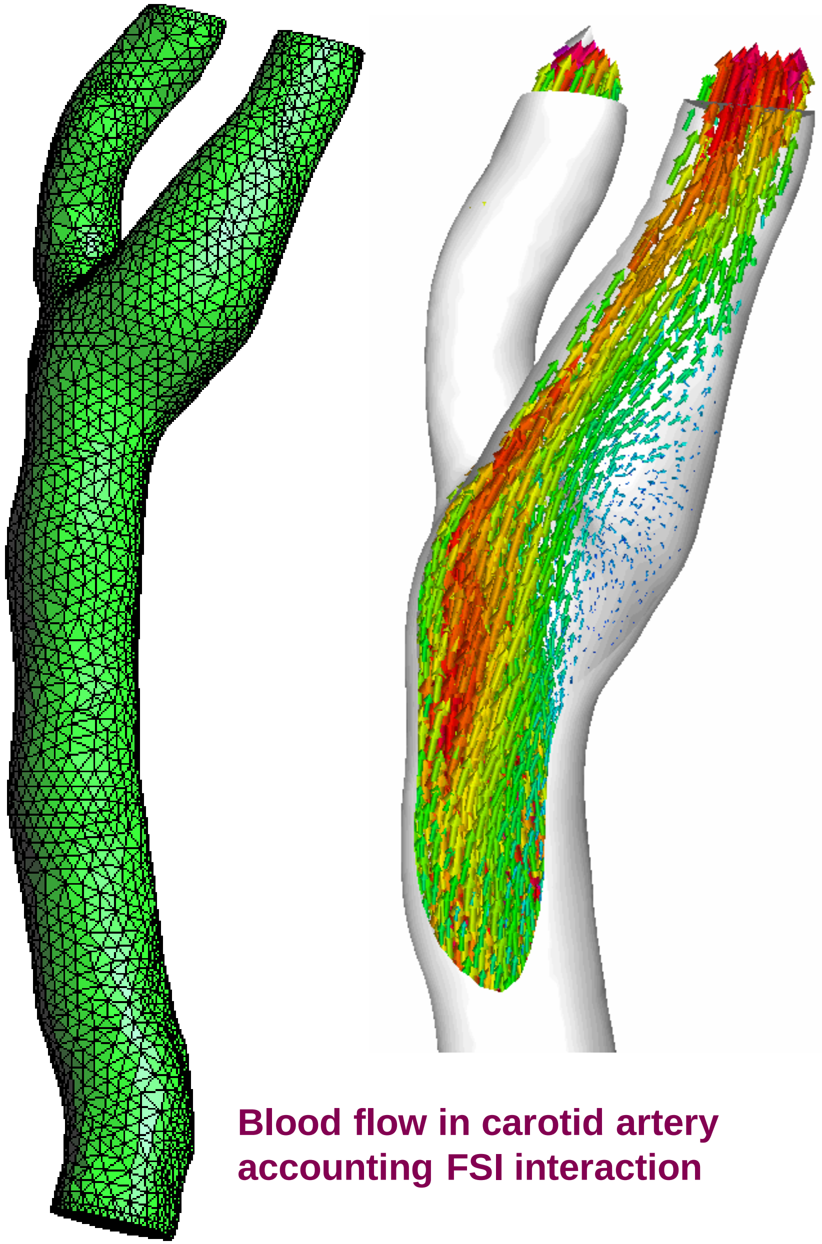
Elems	Dofs	#procs	Time (s)
34^3	171,500	16	44.2
43^3	340,736	32	60.3
54^3	665,500	64	66.7
68^3	1,314,036	128	73.6
86^3	2,634,012	256	83.5
108^3	5,180,116	512	102.0
132^3	9,410,548	1024	106.8

Velocity solves with Hypre: CG + BoomerAMG preconditioner for the 3D driven-cavity case (Re=100) on Cray XC (Sisu).
Simulation Mika Malinen, CSC.

O(~1.14)

Computational Hemodynamics

- Cardiovascular diseases are the leading cause of deaths in western countries
- Calcification reduces elasticity of arteries
- Modeling of blood flow poses a challenging case of fluid-structure-interaction
- Artificial compressibility is used to enhance the convergence of FSI coupling

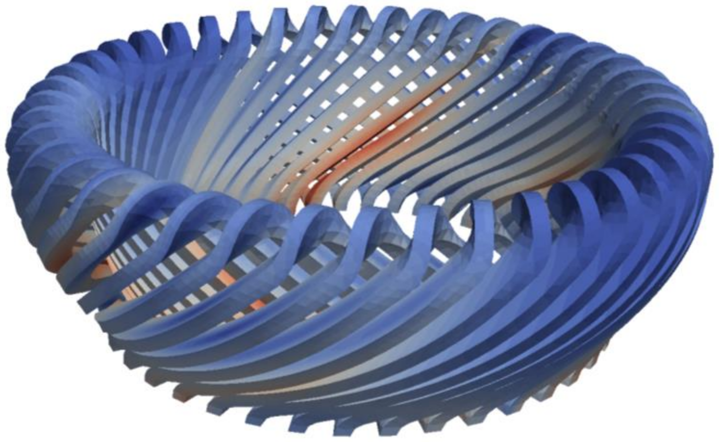


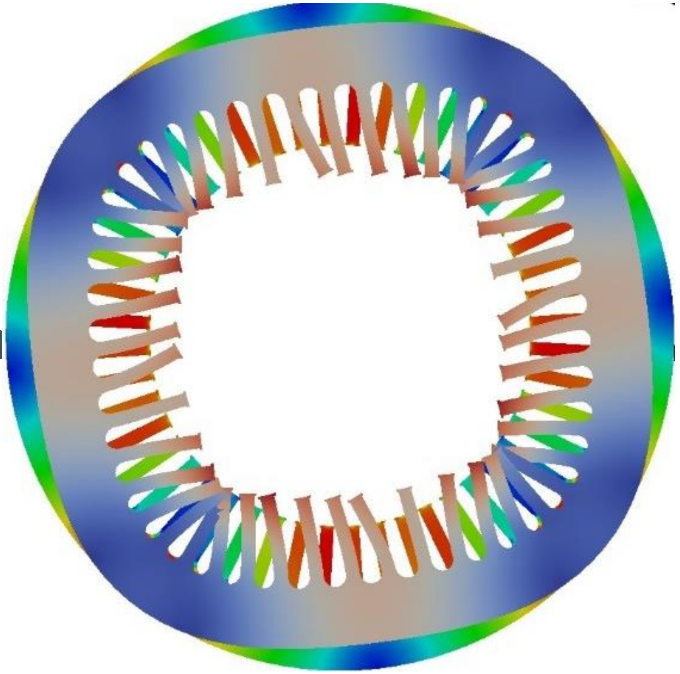
Blood flow in carotid artery accounting FSI interaction

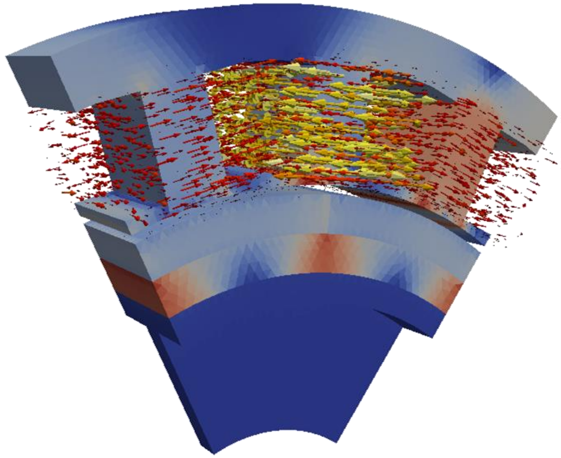
E. Järvinen, P. Råback, M. Lyly, J. Salonius. *A method for partitioned fluid-structure interaction computation of flow in arteries. Medical Eng. & Physics*, **30** (2008), 917-923

Elmer/EM: Collaboration in electromechanics

- SEMTEC project to further develop Elmer as a tool for heavy electromagnetics computations.
  - Existing solution provided unsatisfactory scalability
  - CSC, VTT, Aalto Univ., TUT, LUT, ABB, Kone, Konecranes, Sulzer, Ingersoll-Rand, Trafotek, Scanveir
- With the end of the project large developments made available under open source
- Most important industrial application area at the moment

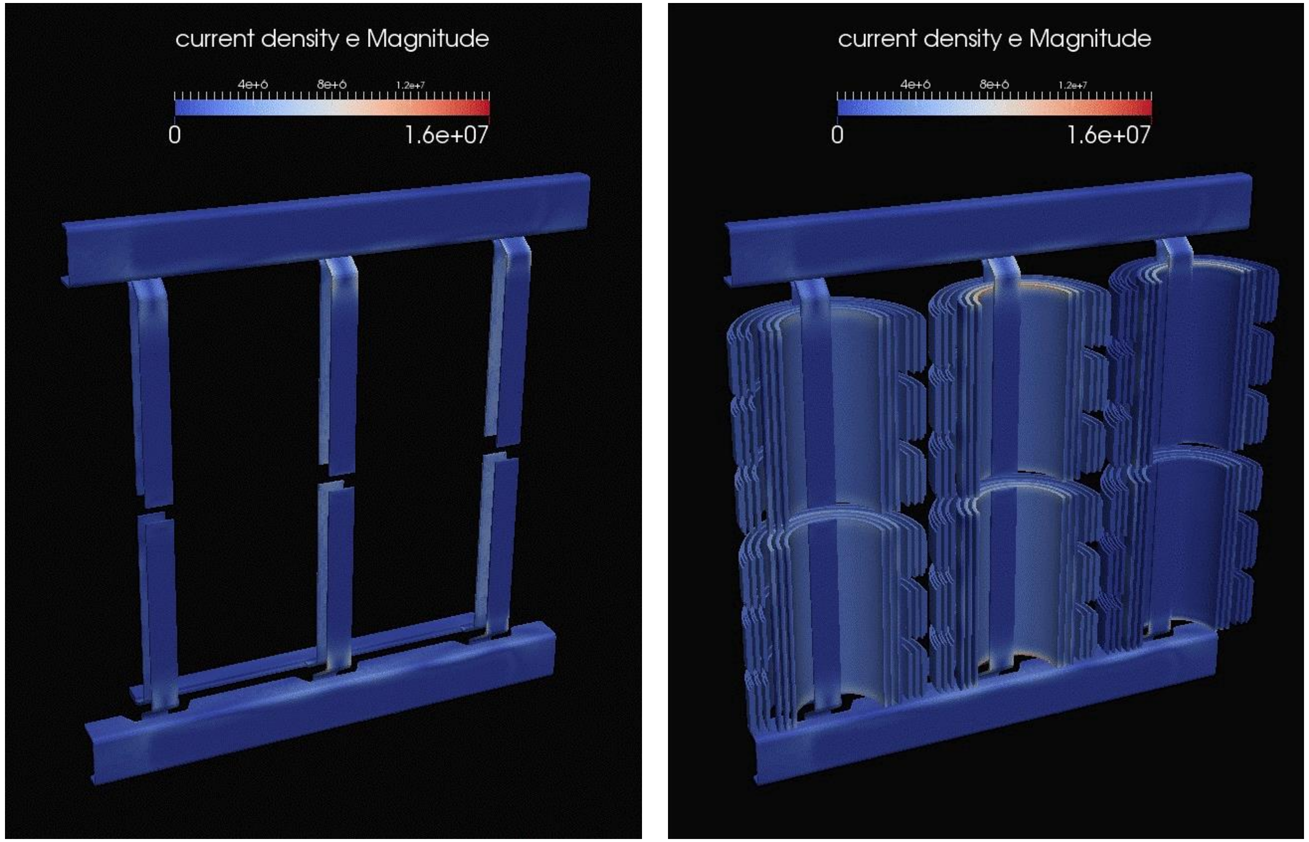






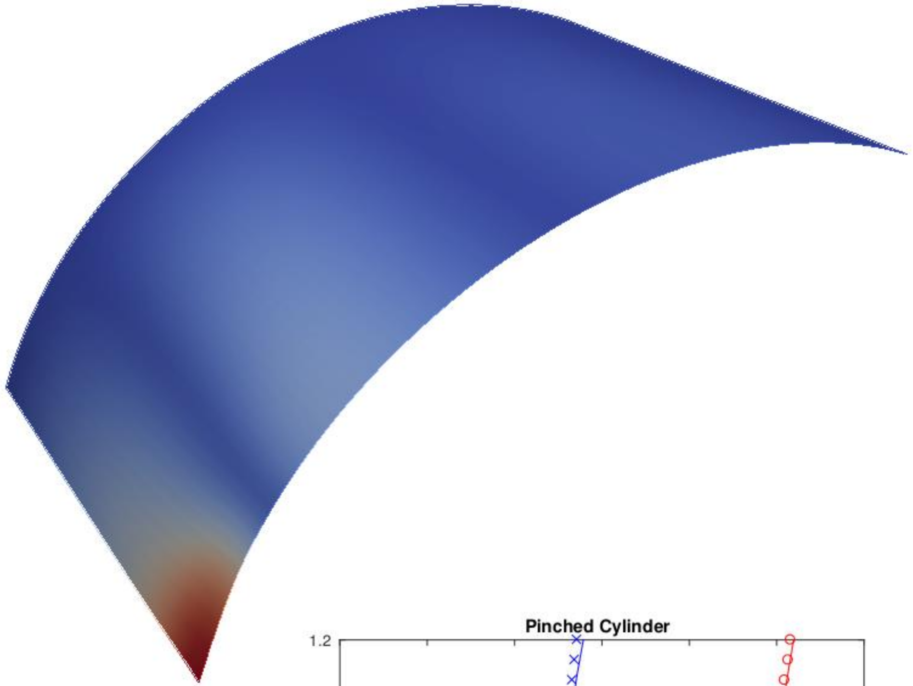
Open source workflow at Trafotek

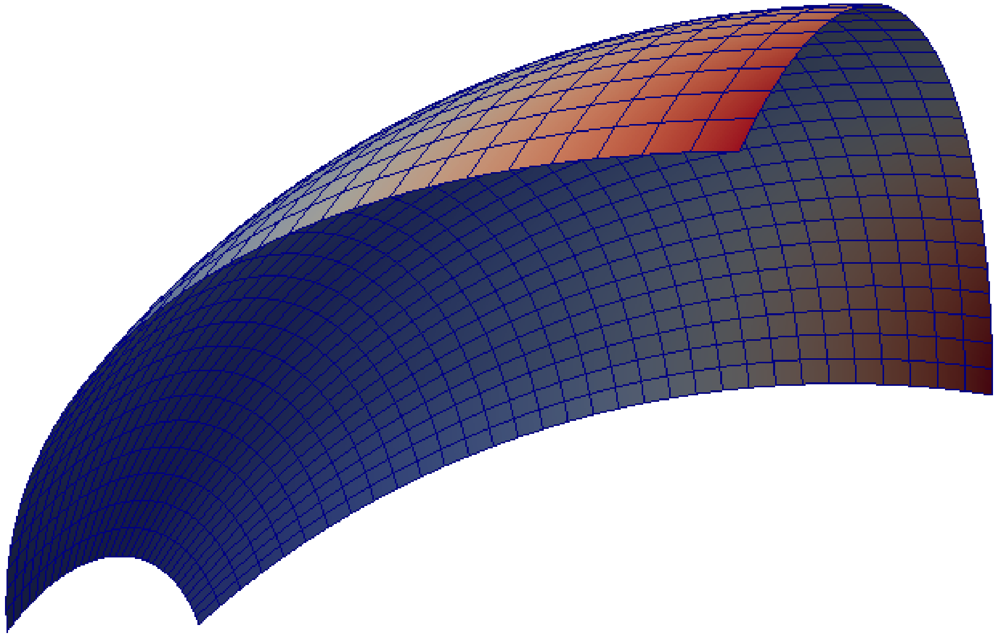
- Simulation of losses in Cast Resin Transformer by Trafotek
  - Computed with 256 cores
- CAD & meshing with **SALOME** using python bindings
- Simulation with **Elmer**
  - Estimation of heat generation from magnetic losses
  - Coupled heat and N-S equations
- Postprocessing with **Paraview**

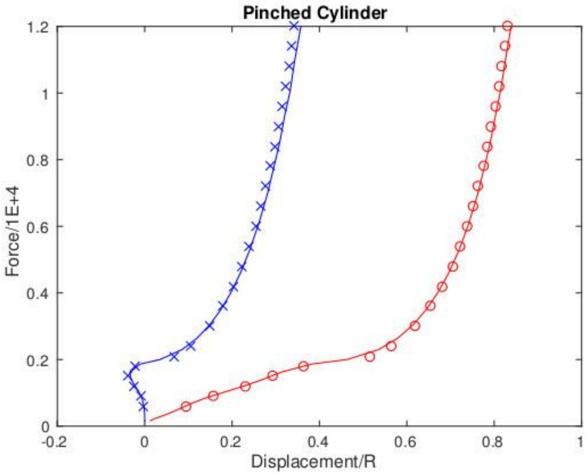


Simulation by Eelis Takala, Trafotek, Finland, 2014

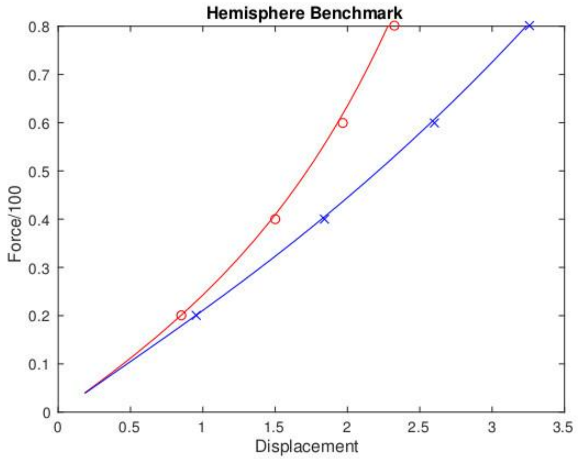
Recent developments: nonlinear shell solver







Verification against:
Sze KY, Liu XH, Lo SH, *Popular benchmark problems for geometric nonlinear analysis of shells*, Finite Elements in Analysis and Design 2004, **40**(11):1551-1569.



Most important Elmer resources

- http://www.csc.fi/elmer
  - Official Homepage of Elmer
- http://www.elmerfem.org
  - Discussion forum, wiki, elmerice community
- https://github.com/elmercsc/elmerfem
  - GIT version control (the future)
- http://youtube.com/elmerfem
  - Youtube channel for Elmer animations
- http://www.nic.funet.fi/pub/sci/physics/elmer/
  - Download repository
- Further information: peter.raback@csc.fi

Thank you for your attention!

ElmerGUI tutorials

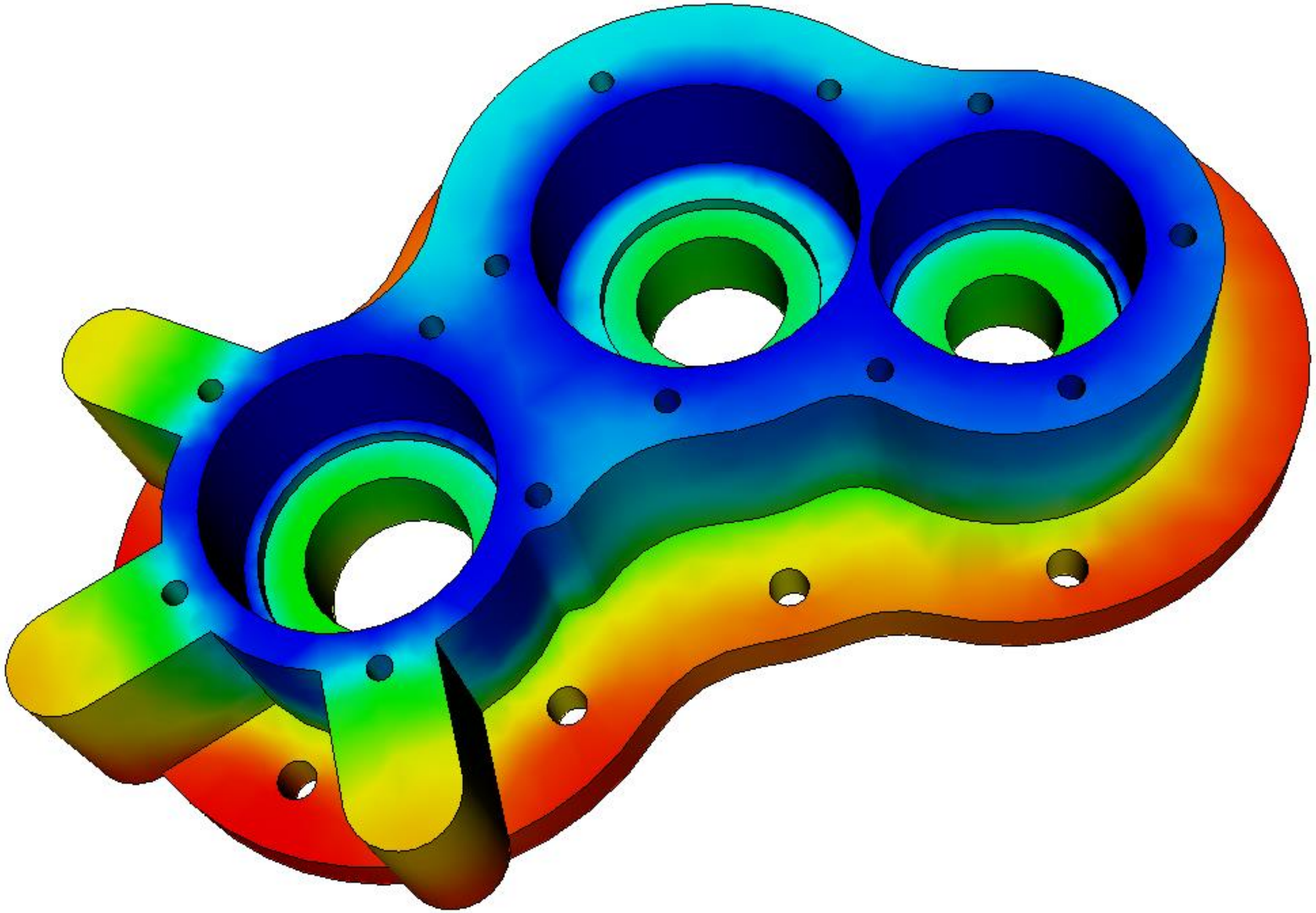
ElmerTeam
CSC – IT Center for Science, Finland

CSC, 2018

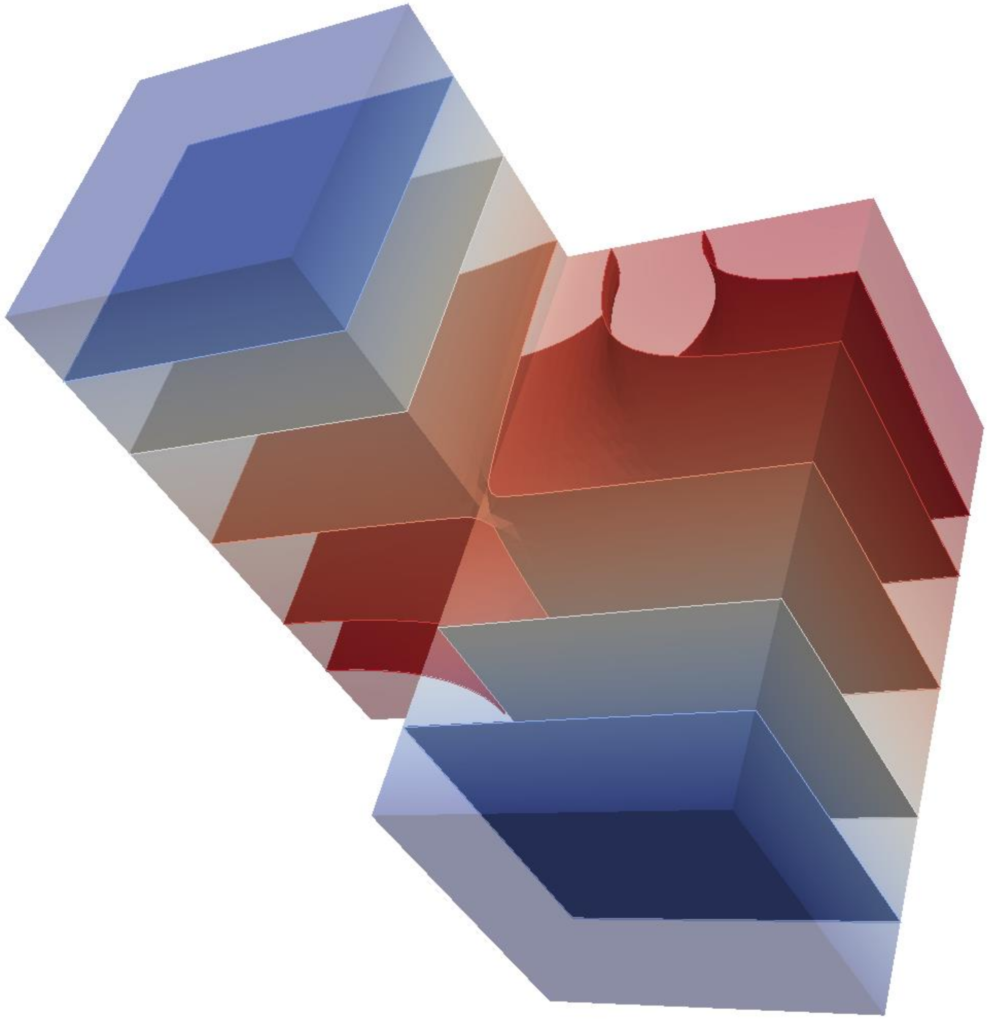
Tutorials GUI instructions

- Copy and unzip the virtual machine
- Start Virtual Machine under VMPlayer
- Input files should be available in Desktop shortcut or under **`~/Source/elmerfem/ElmerGUI/samples`**
- The instructions written in verbatim refer to operations with the GUI.
    - Intendation means step in the menu hierarchy.
- Missing solver menus may be found at **`/usr/local/Elmer_devel/share/ElmerGUI/edf-extra`**
- Use **Paraview** with Post File suffix **`.vtu`**

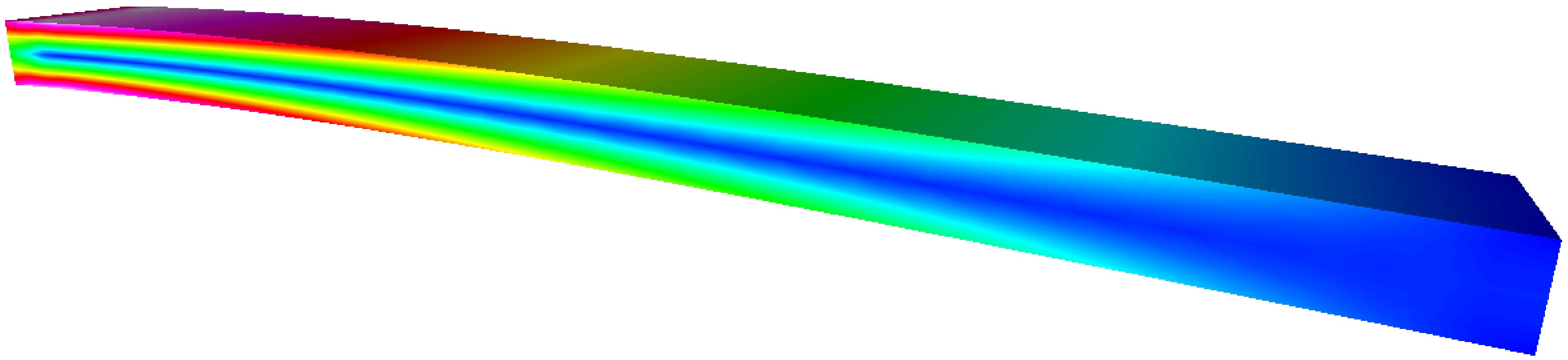
1. Heat Equation - Temperature field of a solid object



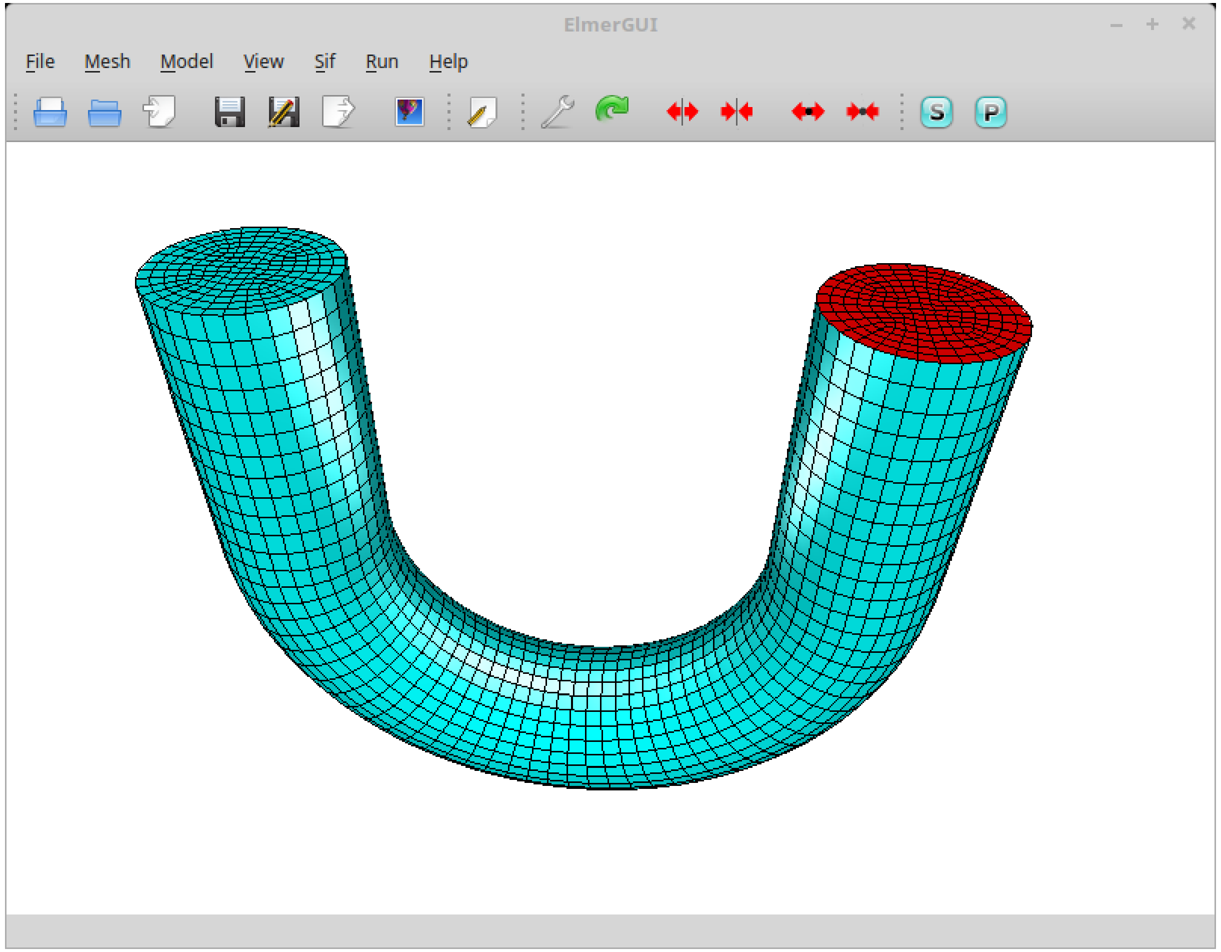
2. Model PDE – generic advection-reaction-diffusion equation

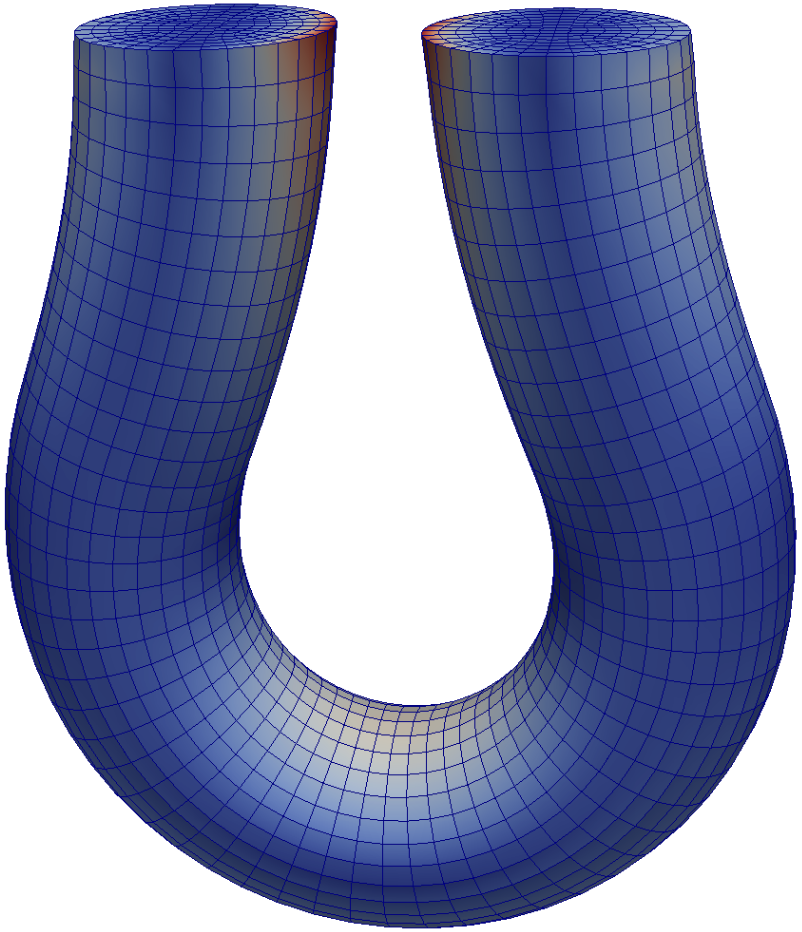


3. Linear elasticty - Loaded elastic beam in 3D

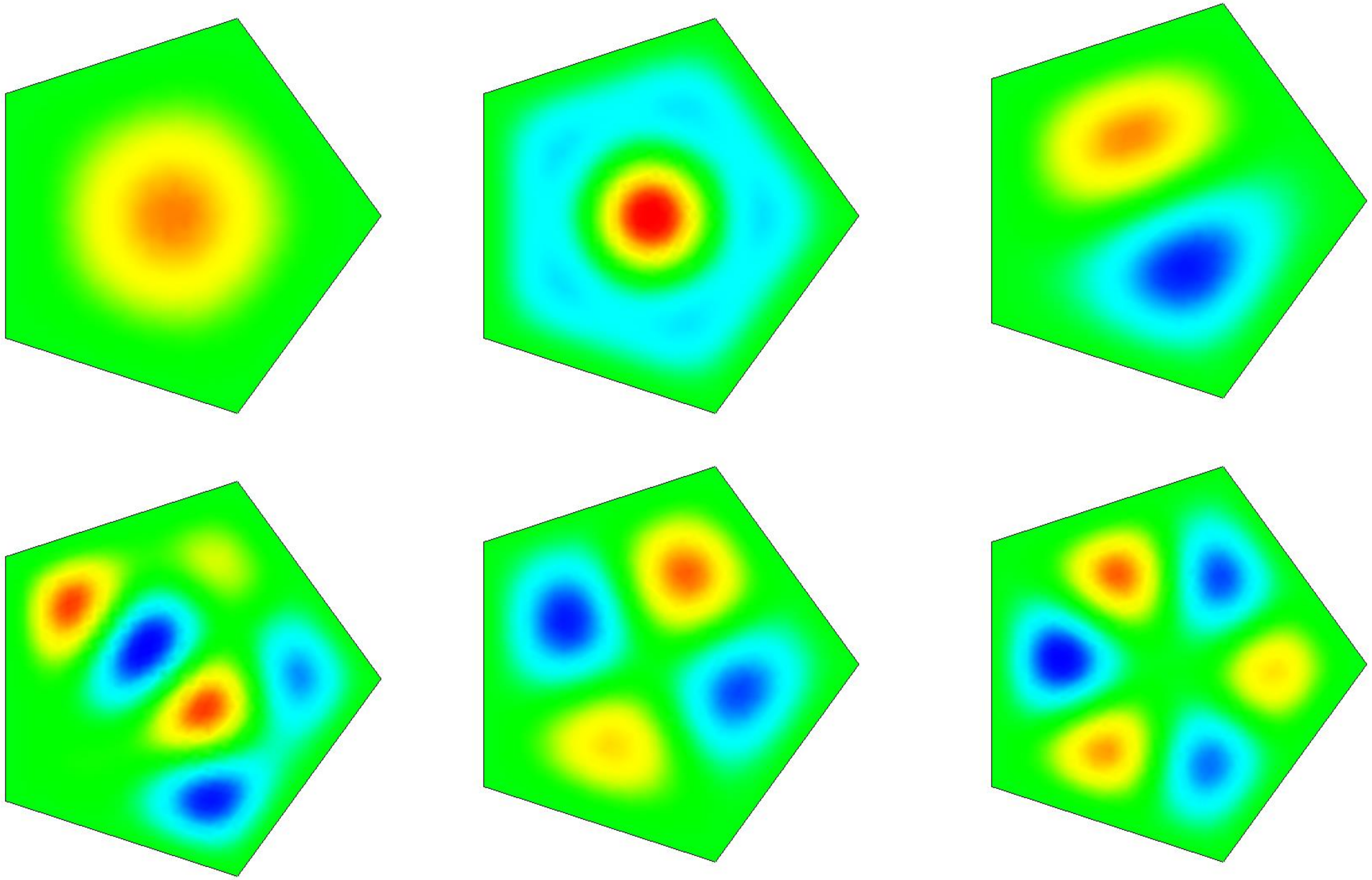


4. Nonlinear elasticity – loaded elastic hook

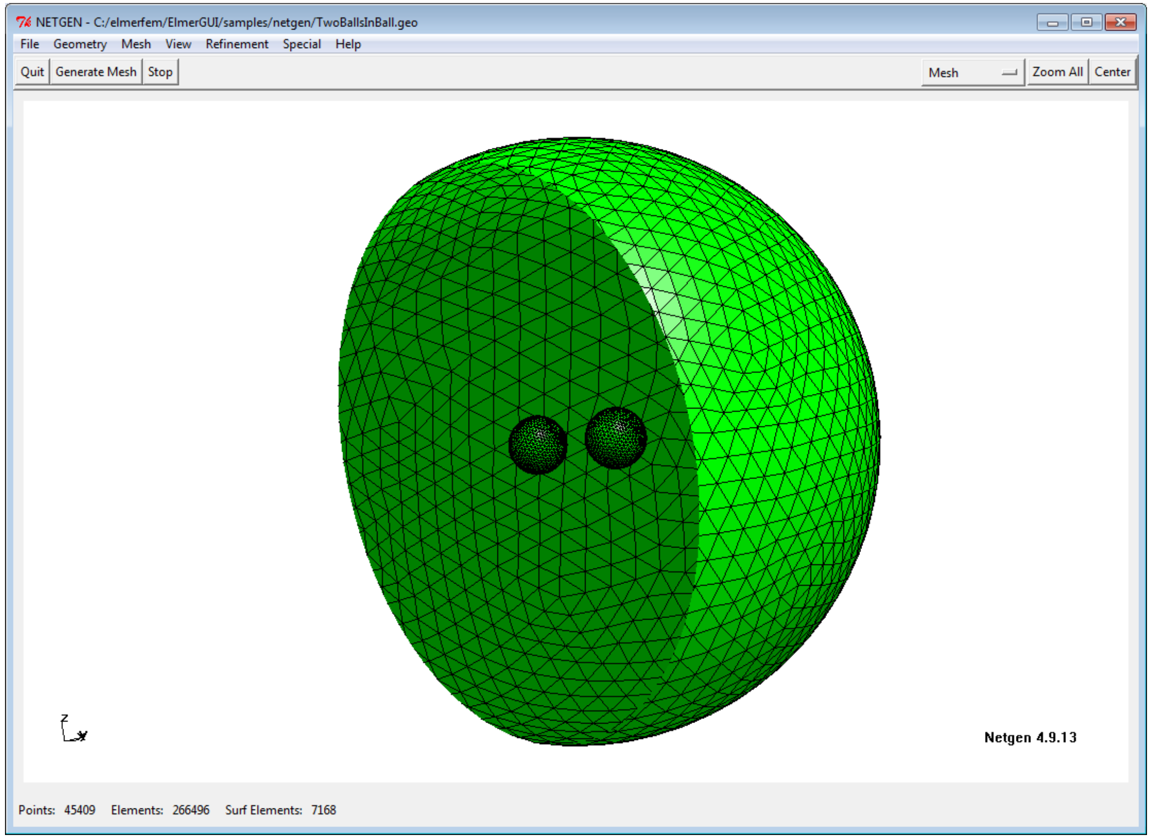


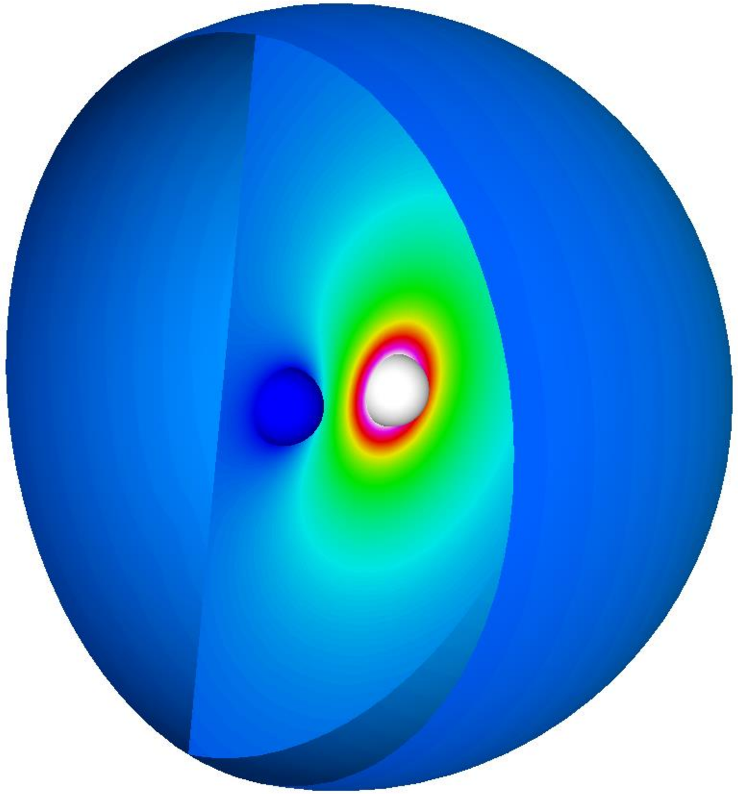


5. SmitC Solver - Eigenmodes of a elastic plate

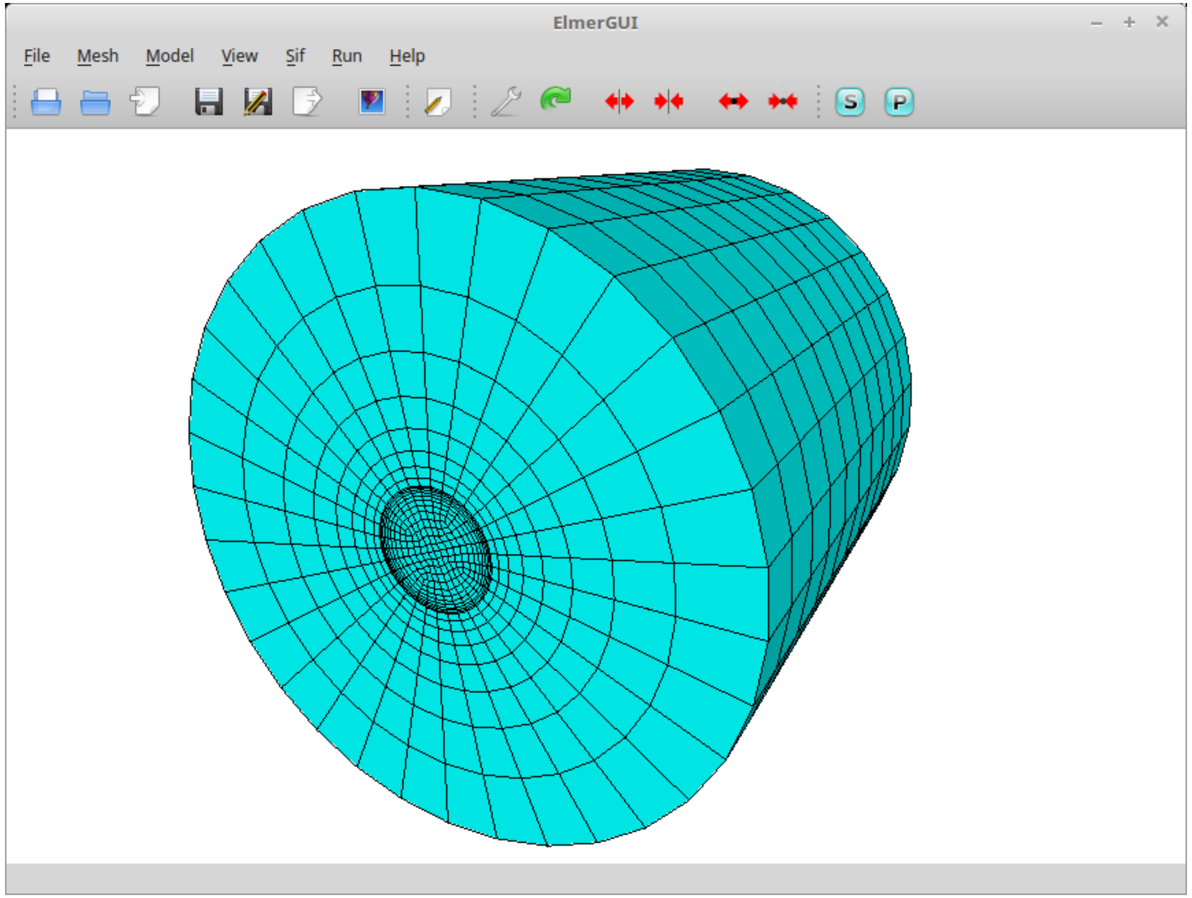


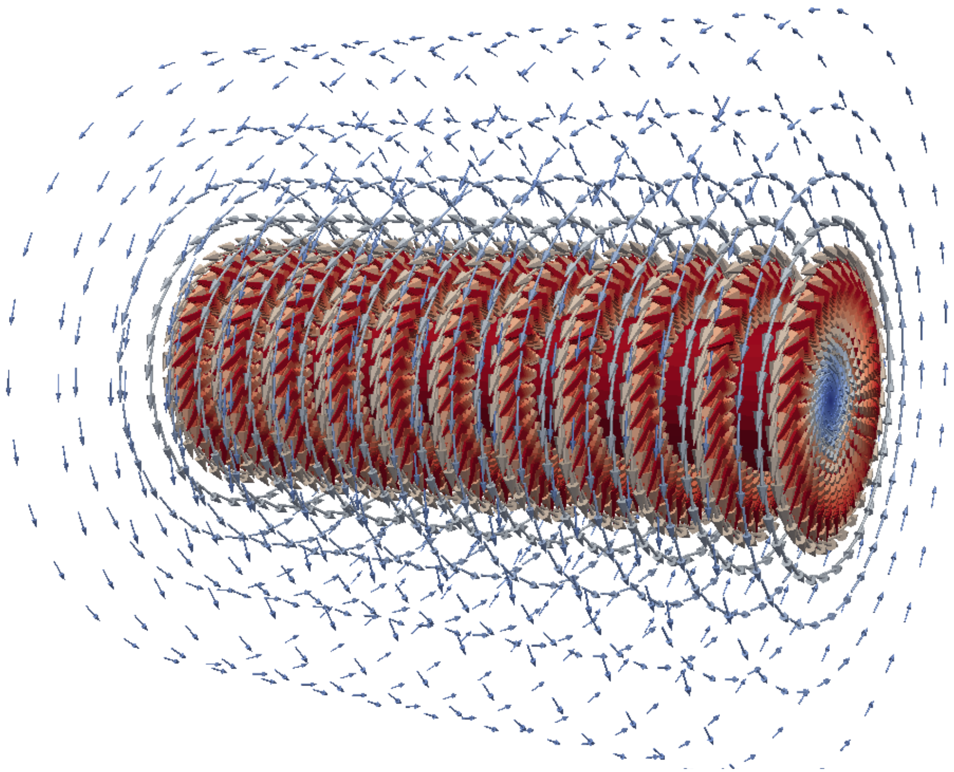
6. Electrostatics solver - Capacitance of two balls in free space

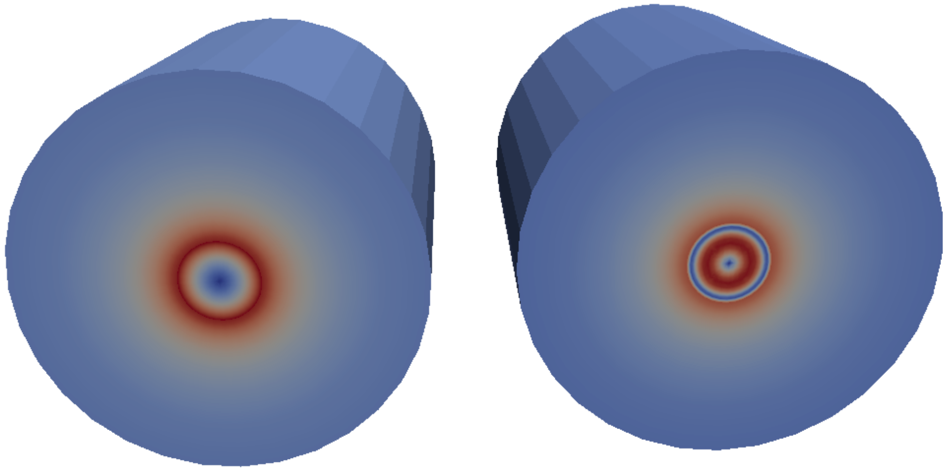




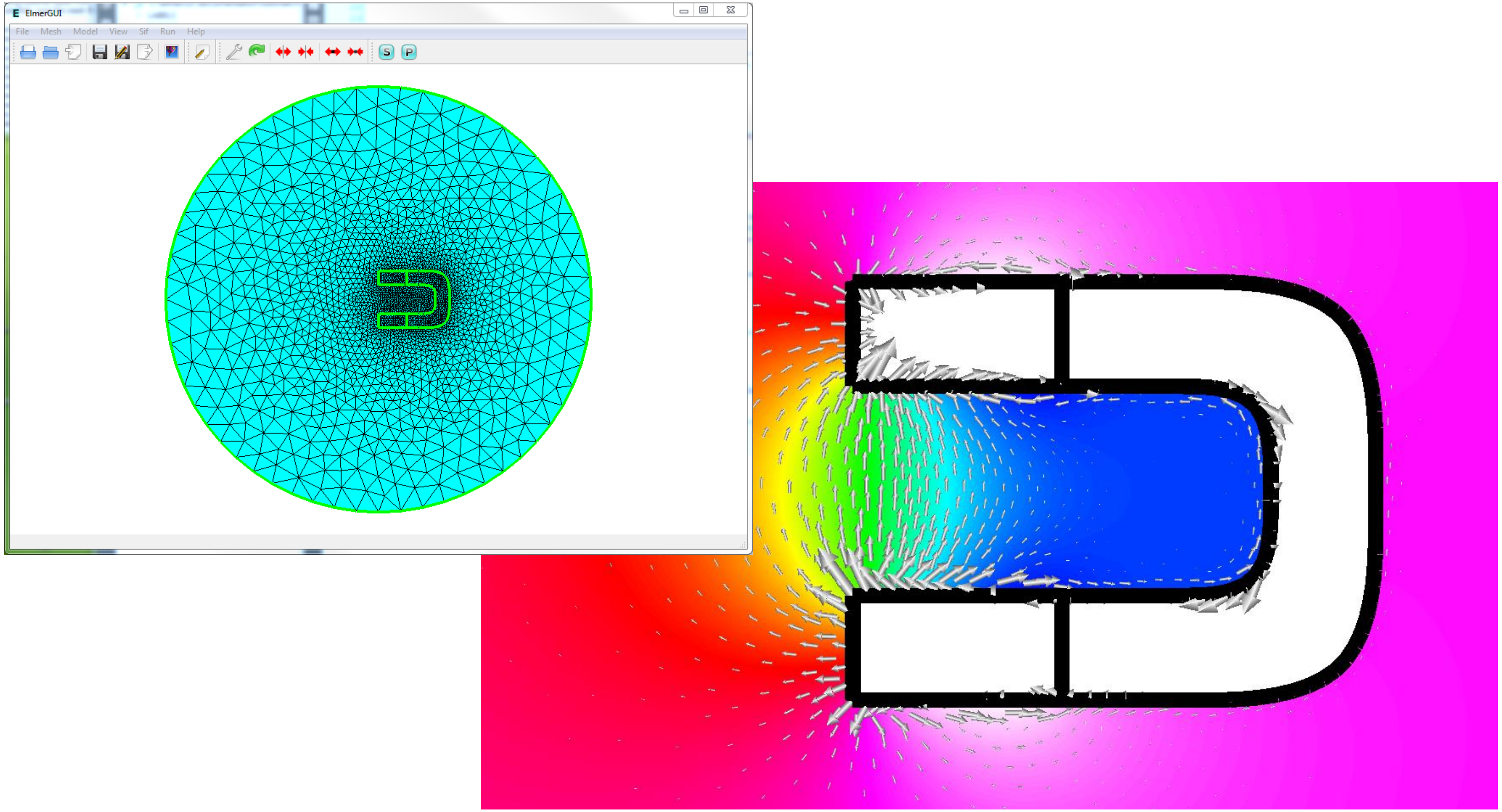
7. AV Solver – Magnetic field around a wire



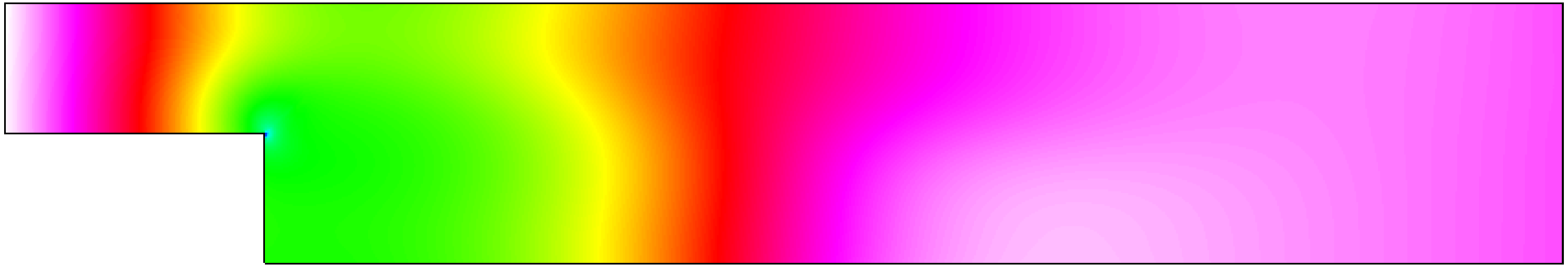


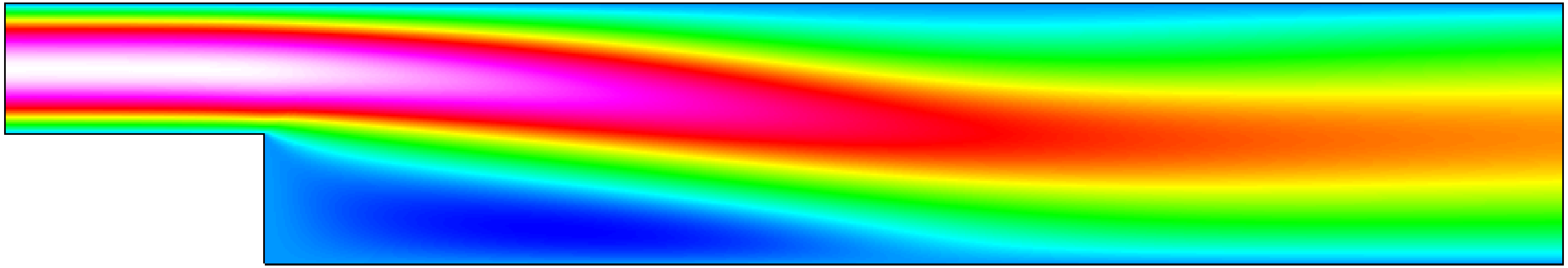


8. MagnetoDynamics2D- Permanent magnet shaped like a horse shoe

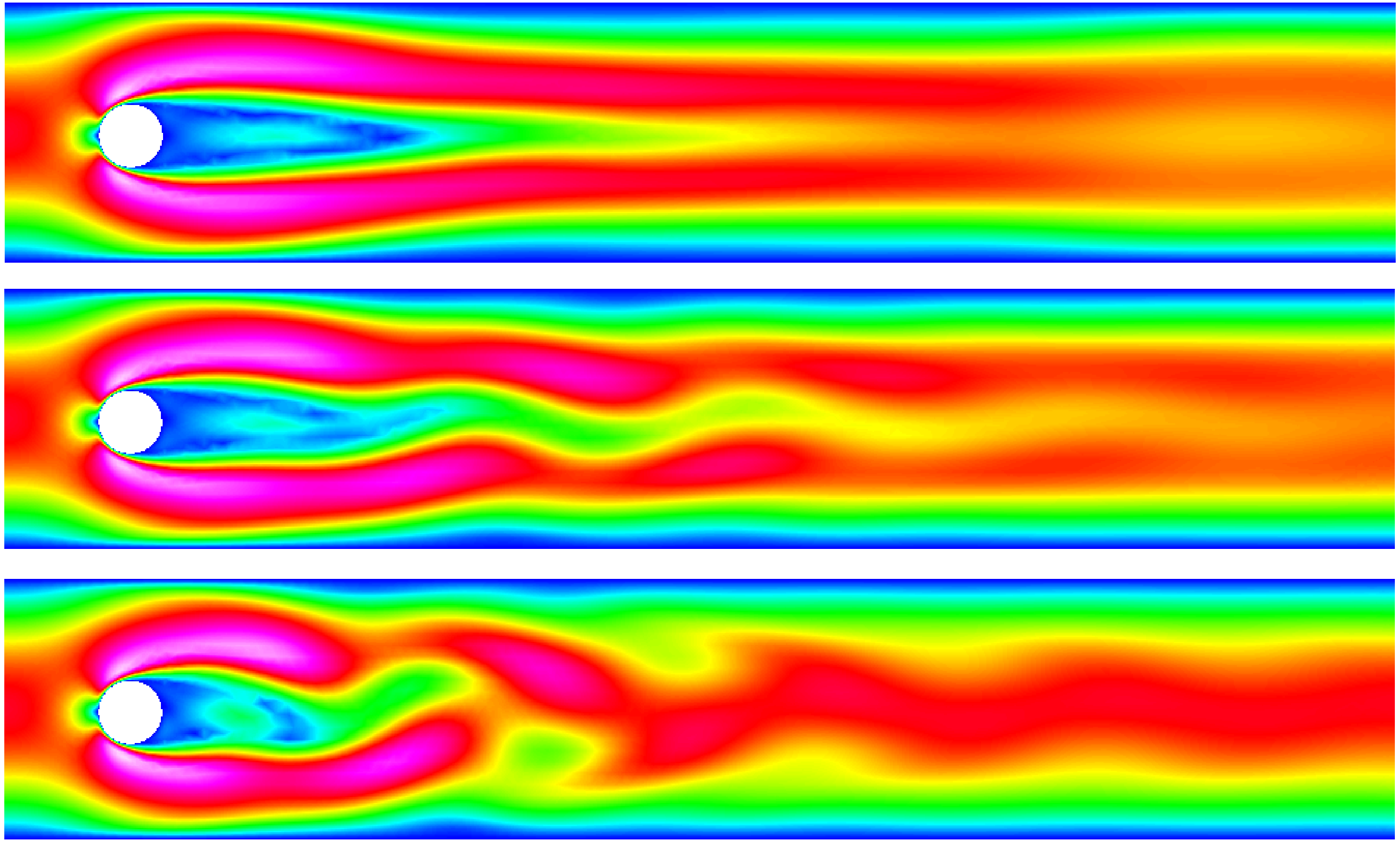


9. Navier-Stokes equation - Flow passing a step, Re=100

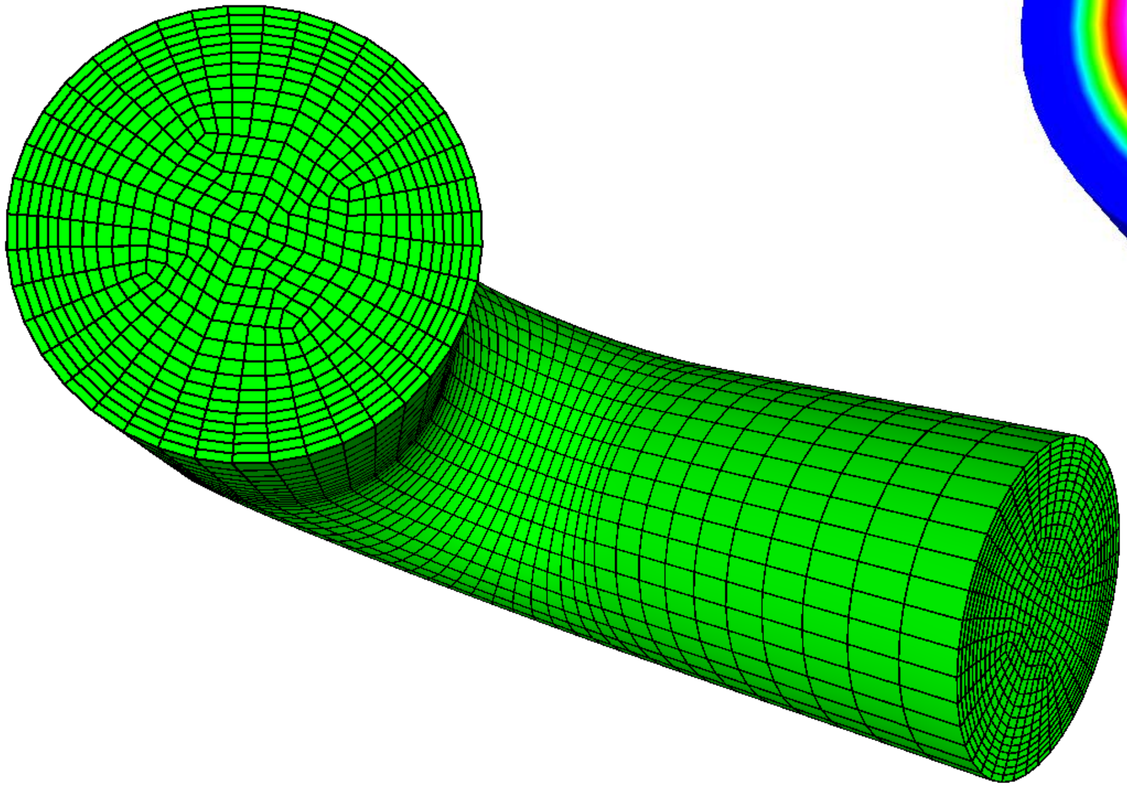


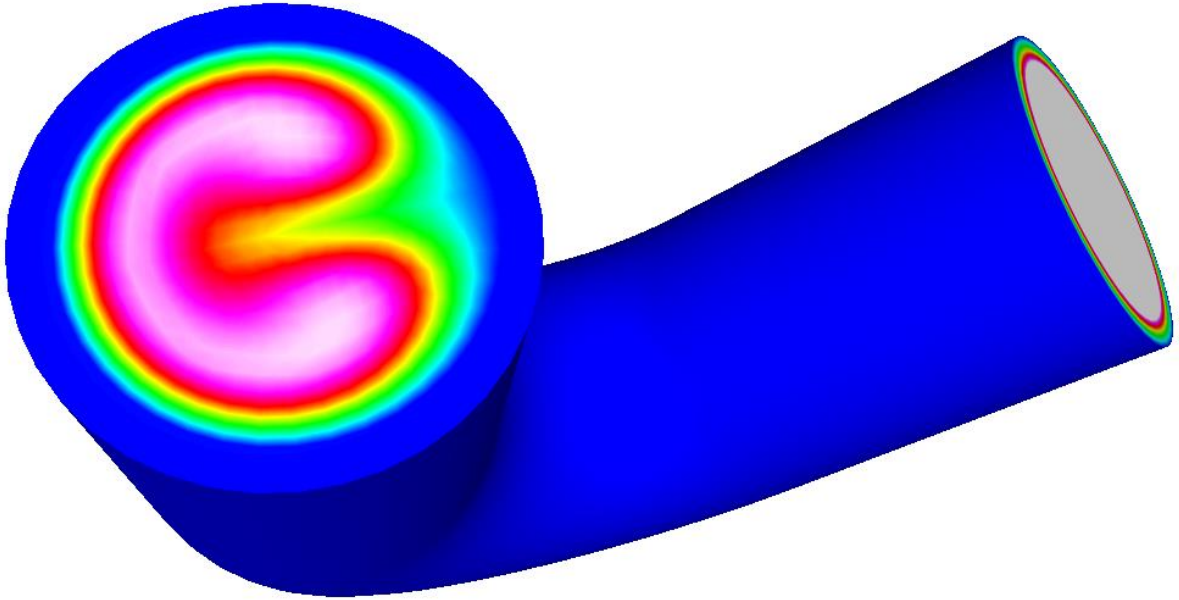


10. Navier-Stokes equation - Vortex shedding

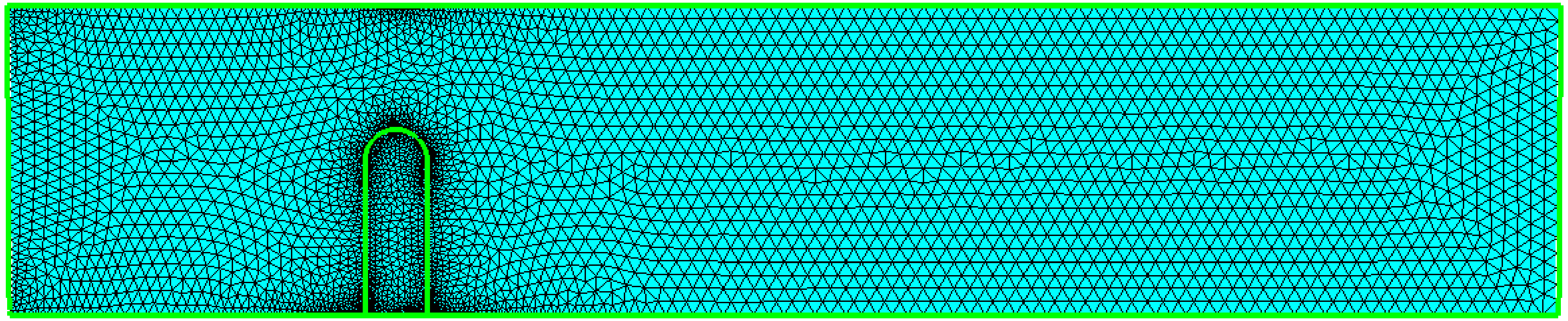


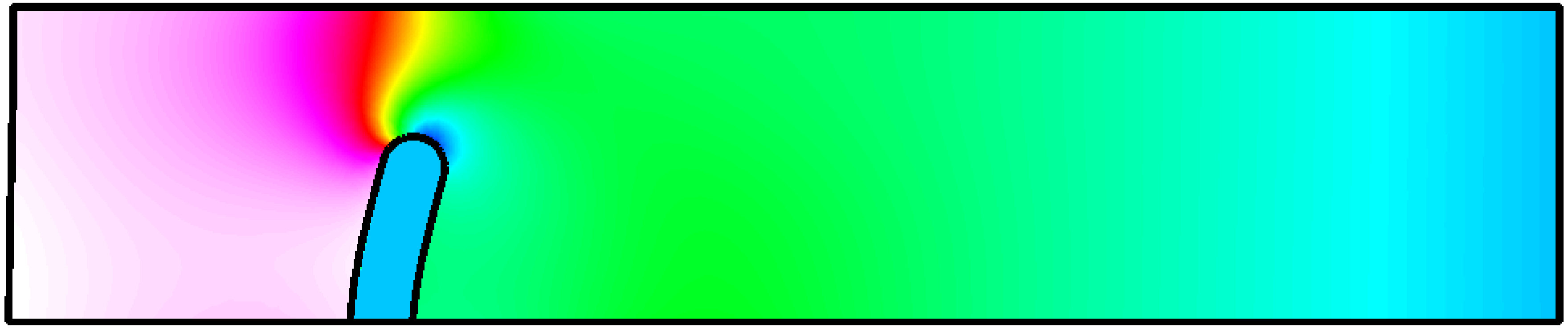
11. Thermal flow in curved pipe



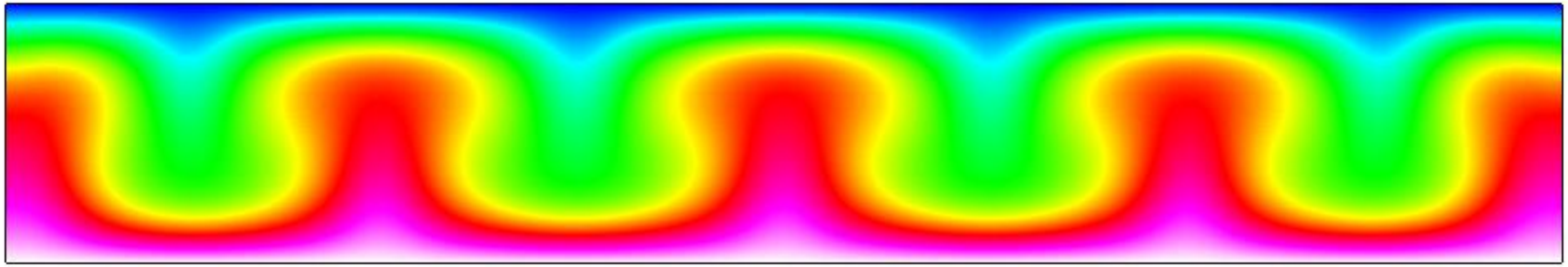


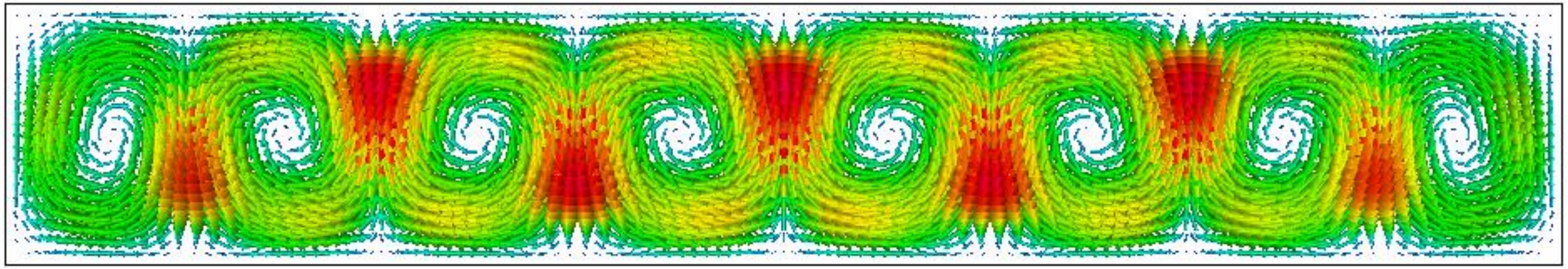
12. FSI - Obstacle with fluid-structure interaction



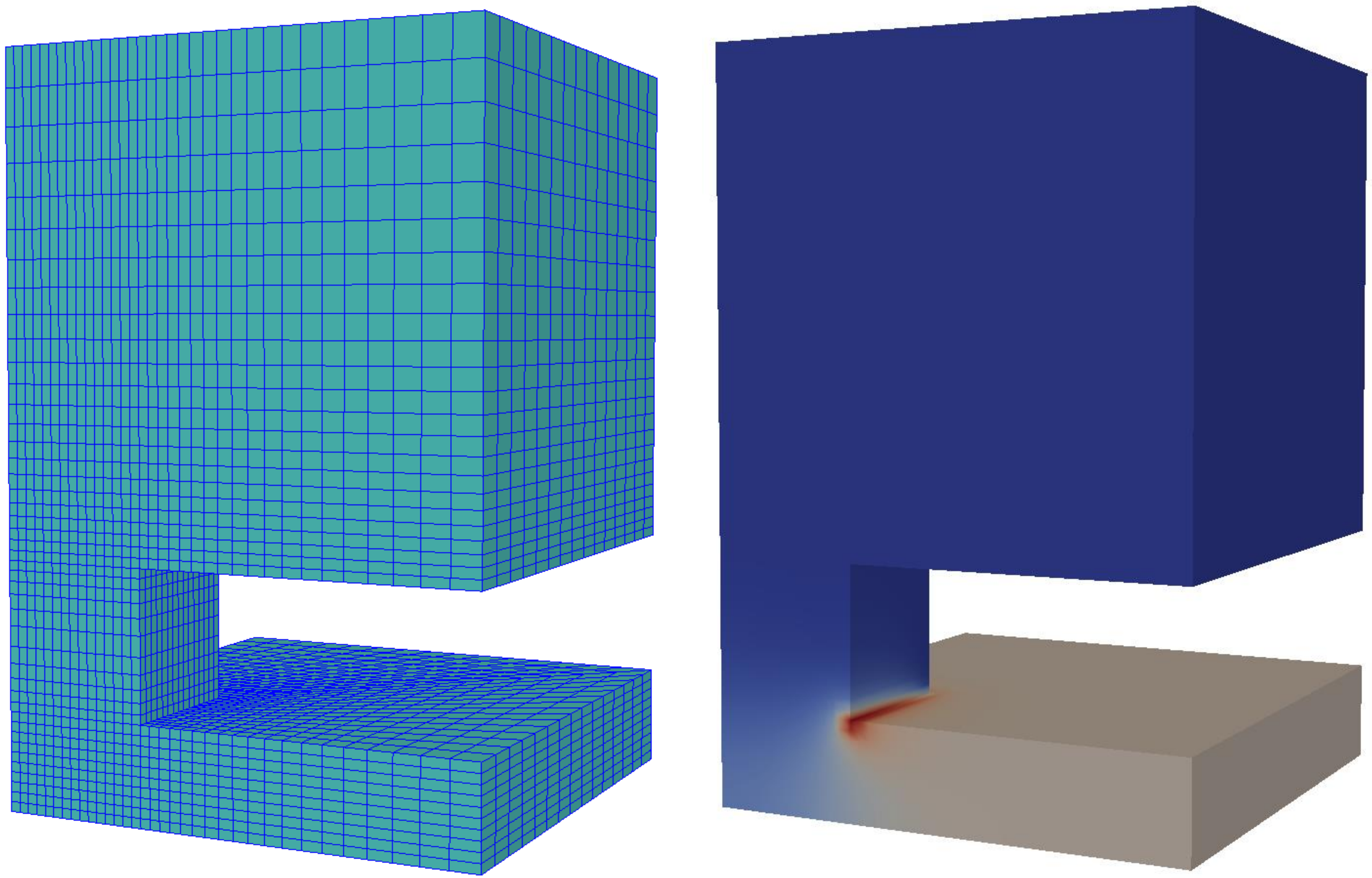


13. Rayleigh-Benard Instability

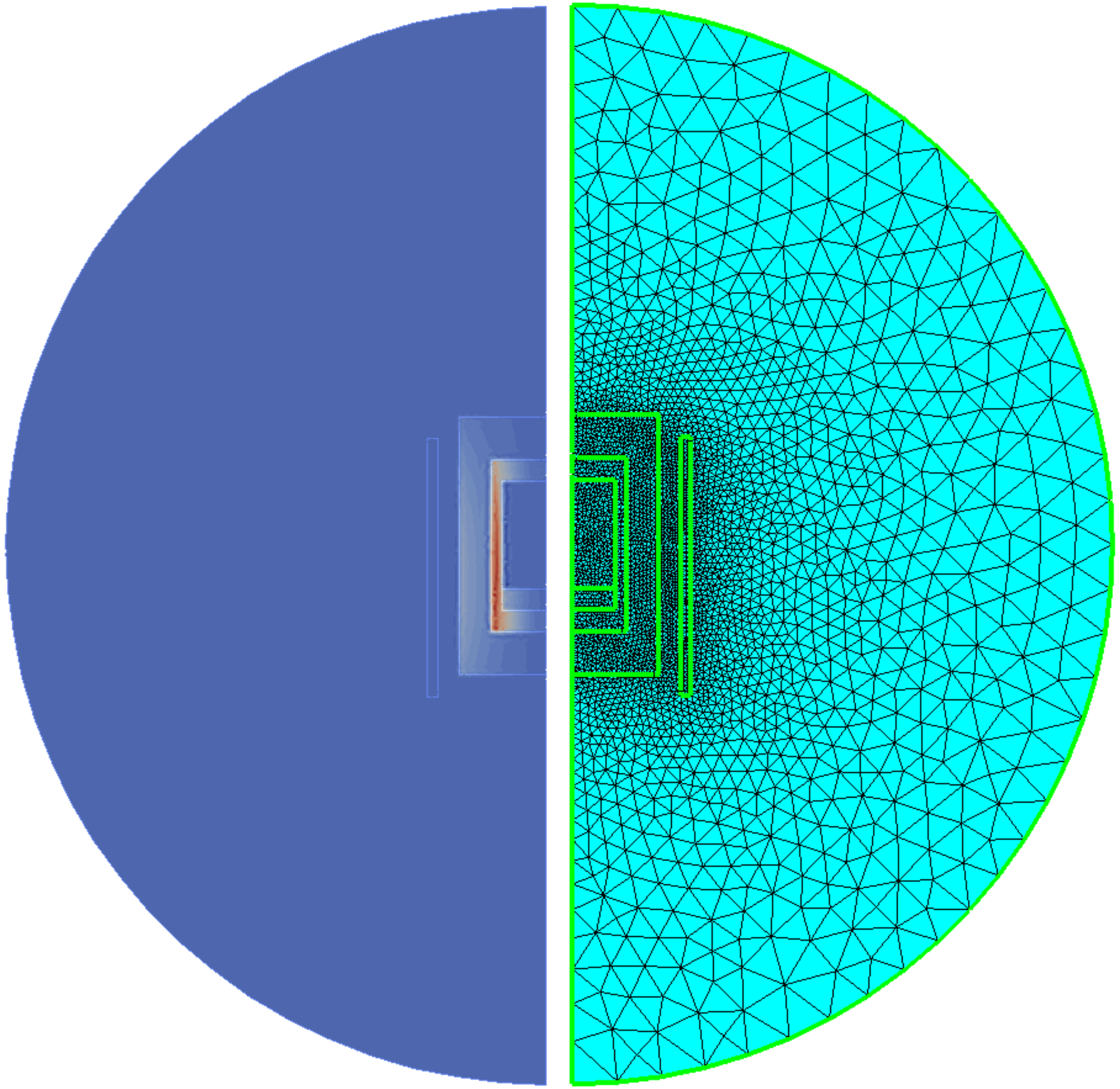




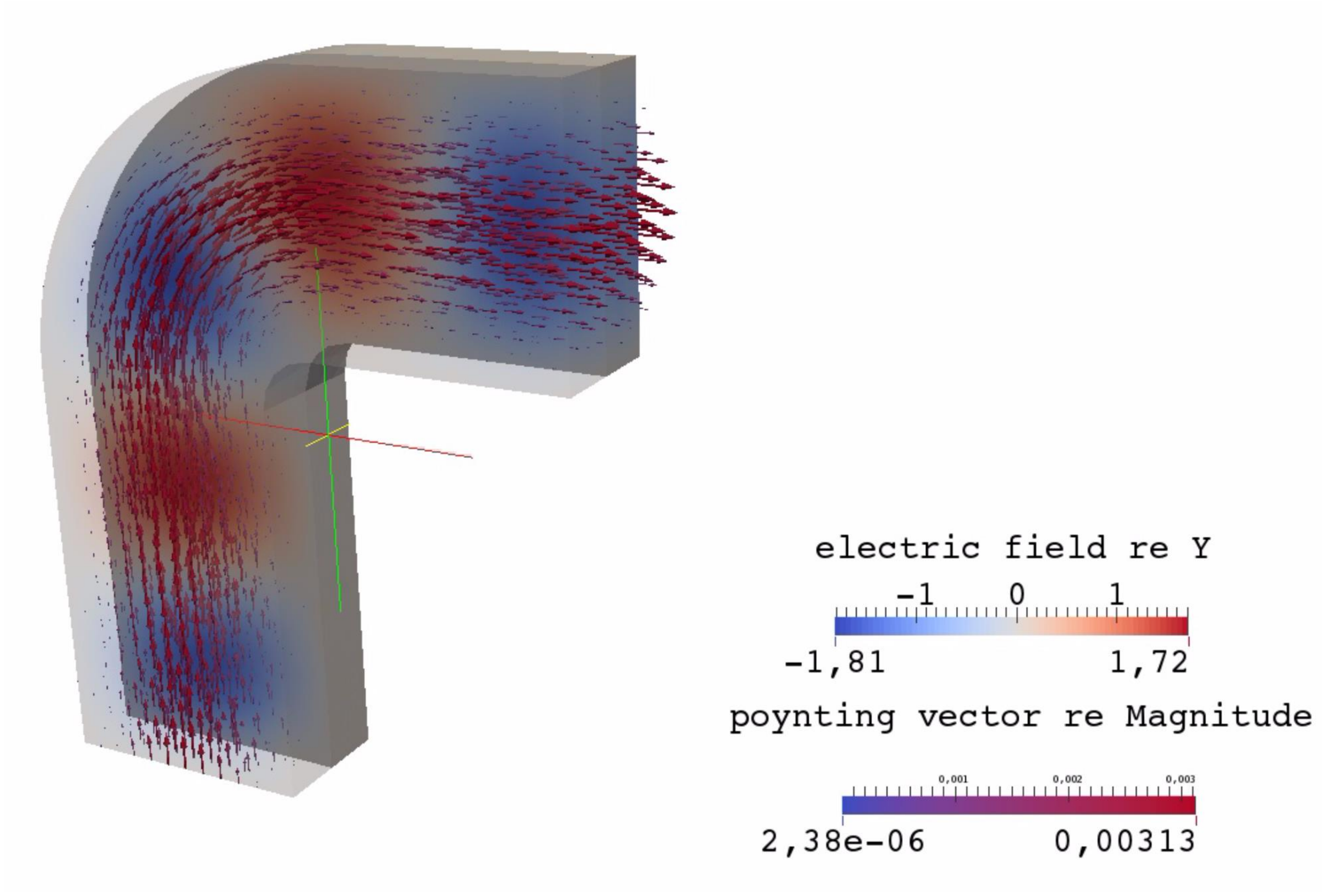
14. Capacitance of a perforated plate



15. Induction heating in a graphite crucible



16. Electromagnetic waves in a waveguide

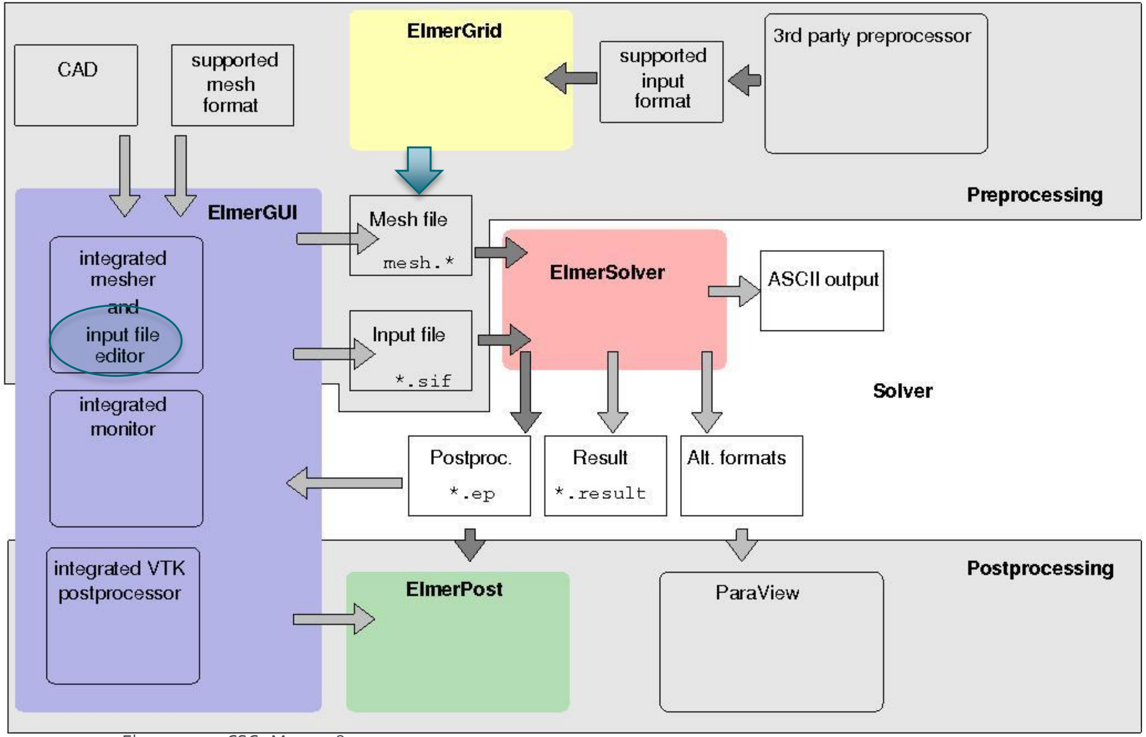


Thermal flow in a curved pipe

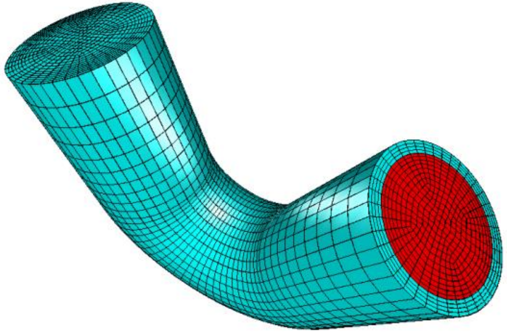
Explaining basic structure of an Elmer simulation

Elmer Team
CSC – IT Center for Science Ltd.

Elmer - Modules



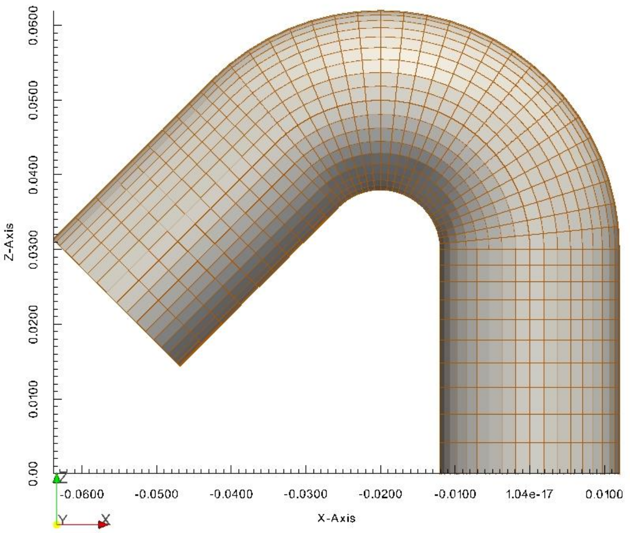
The problem



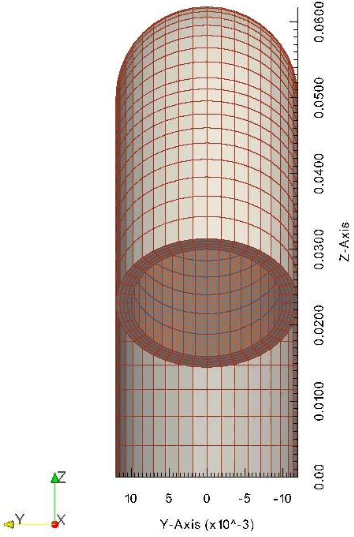
This is the current Tutorial 11
Thermal Flow in a curved pipe
in ElmerTutorials.pdf
(from nic.funet.fi)

- Pipe consisting of solid (iron) wall filled with fluid (water)
- We have a hot (350 K) inflow on one side of the pipe and cool the outside of the pipe at 300 K
- We prescribe inflow profile of water
- We are interested in steady state solution

The problem



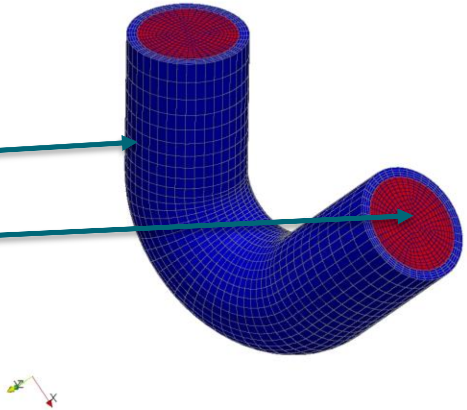
0.00
0.0100
0.0200
0.0300
0.0400
0.0500
0.0600
Z-Axis
-0.0600
-0.0500
-0.0400
-0.0300
-0.0200
-0.0100
1.04e-17
0.0100
X-Axis
Z
Y
X



Z
Y
X
10
5
0
-5
-10
Y-Axis (x10^-3)
0.00
0.0100
0.0200
0.0300
0.0400
0.0500
0.0600
Z-Axis

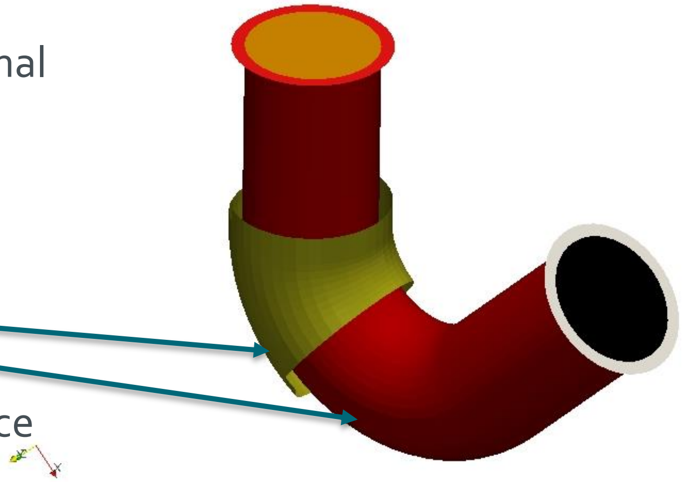
On bodies

- A **Body** is a distinguishable part of the computational domain
  - Geometry
  - Physical model(s)
  - Material properties
- Here we have two bodies, because we have two different material (+ different physical models)
  - Solid (iron): heat transfer
  - Fluid (water): flow + heat transfer



On boundaries

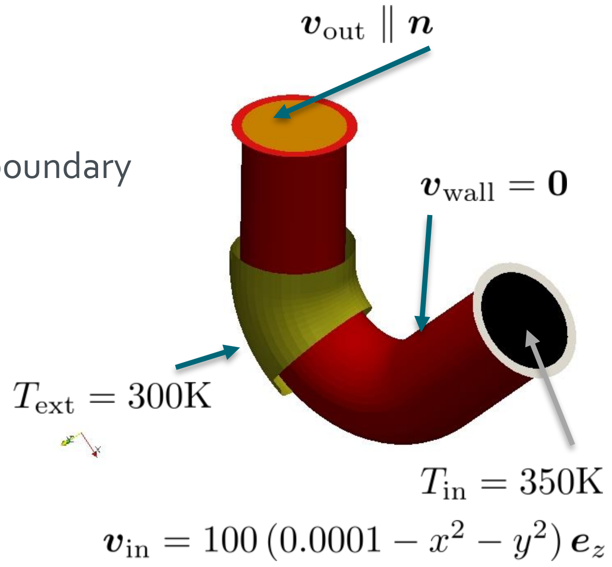
- A **Boundary** is a distinguishable lower-dimensional entity of the computational domain
  - In 3D: surfaces, lines and nodes
  - In 2D: lines and nodes
  - Can confine a body (external)
  - Can be situated in between 2 bodies (internal)
- Here we have several outside- and internal surface **boundaries**
  - Can be viewed with ParaView



On boundaries

- A **Boundary Condition** is a set of instructions that declares
  - values of variables (**Dirichlet-condition**) or their normal
  - gradients (**Neumann-condition**) or mixed (**Robin-condition**) on a boundary
- Mind: BC's can apply to multiple boundaries
  - Don't interchange boundary with boundary condition

Suggestion: if you want to, you can start a little bit easier by just imposing a constant inflow velocity of 0.01



On solvers

- We talk of **Solvers** in terms of different physical models formulated by PDE's

- Heat transfer

$$\rho\, c\,(\partial T/\partial t + \boldsymbol{u}\cdot\nabla T) = \nabla\cdot(\kappa\nabla T) + \rho\sigma$$

- Navier-Stokes

$$\rho\,(\partial \boldsymbol{u}/\partial t + \boldsymbol{u}\cdot\nabla \boldsymbol{u}) = -\nabla\, p + \nabla\cdot(\mu\dot{\boldsymbol{\epsilon}}(\boldsymbol{u})) + \rho\boldsymbol{f}$$

Material

- A **Material** defines the physical parameters

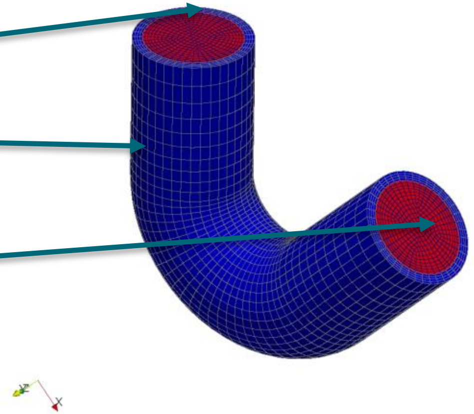
- Heat transfer

$\rho,\ c,\ \kappa$

- Navier-Stokes

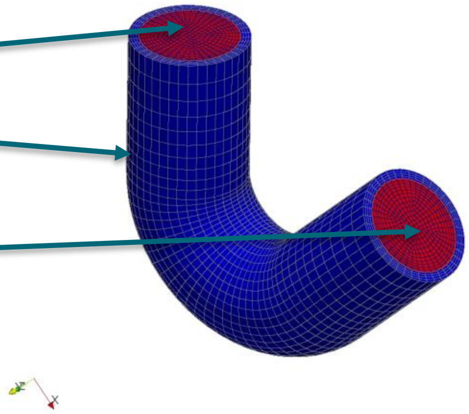
$\rho,\ \mu$

- In our case we used material library in GUI



Bodyforce

- A **Body Force** defines the right-hand side of the equations
- Heat transfer
  - Heat source
- Navier-Stokes
  - Flow Body Force
  - Just theoretical, as we do not apply in this case



Equation

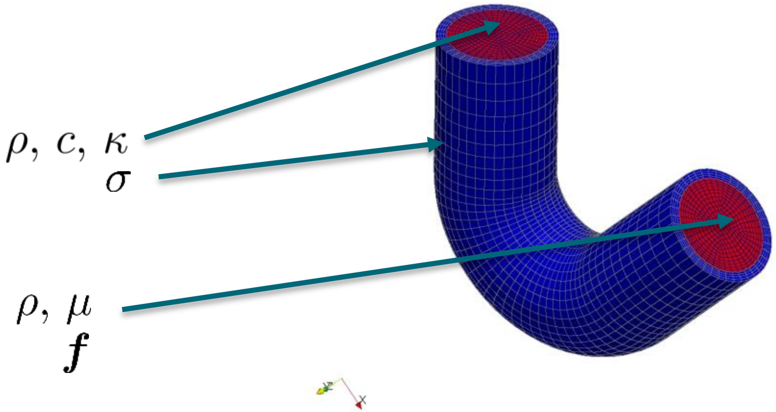
- An **Equation** assigns the solvers/materials/body forces to the different bodies

- Heat transfer

$$\rho\, c\, (\partial T/\partial t + \boldsymbol{u}\cdot\nabla T) = \\ \nabla\cdot(\kappa\nabla T) + \rho\sigma$$

- Navier-Stokes

$$\rho\,(\partial \boldsymbol{u}/\partial t + \boldsymbol{u}\cdot\nabla \boldsymbol{u}) = \\ -\nabla p + \nabla\cdot(\mu\dot{\boldsymbol{\epsilon}}(\boldsymbol{u})) + \rho\boldsymbol{f}$$

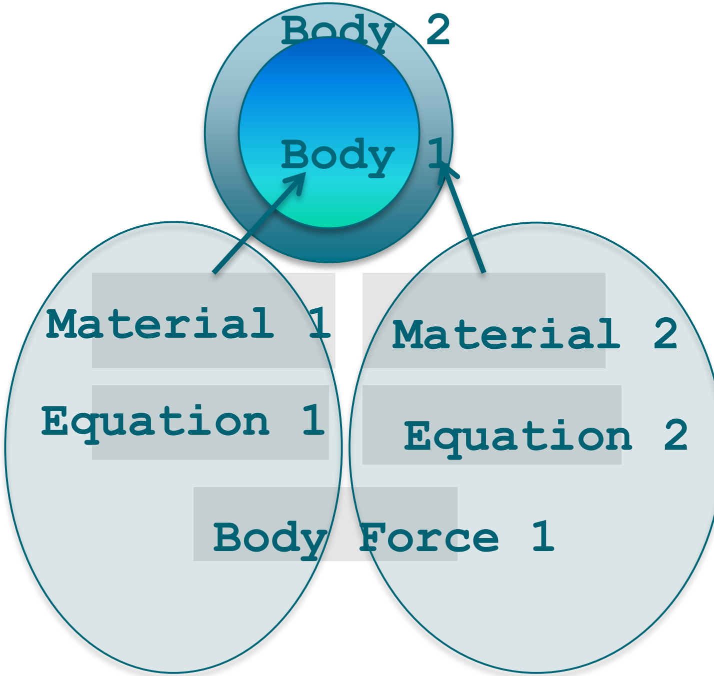


Equation

- Each **Body** has to have an **Equation** and **Material** assigned
- **Body Force**, **Initial Condition** are optional
- Two bodies can have the same

  Material/Equation/

  Body Force/Initial Condition section assigned



Further settings to change

- Setup
  - Change **`case.ep`** into **`case.vtu`** in order to obtain output for ParaView
  - For restart, type into Free text input field:

```
Output File = case.result
```

- Equation
  - Heat and Flow
  - Tab: Heat Equation
  - Edit Solver Settings
  - The Material parameters for heat transfer are constant. Hence this is a linear problem in terms of the variable Temperature:

```
Nonlinear System Max
Iterations = 20 → 1
```

Thermal flow in a curved pipe

Variations on the tutorial case using modifications of the text input file: coupling, MATC, User Defined Functions

Elmer Team
CSC – IT Center for Science Ltd.

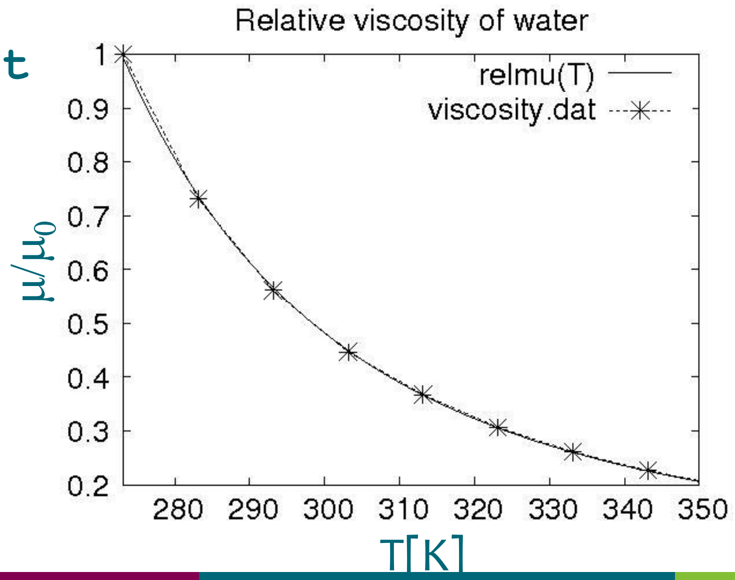
Variations – 2 way coupling

- Temperature dependence of the viscosity for liquid water

$$\mu/\mu_0 = \exp\{-1.704 \ - 5.306\frac{273.15}{T} + 7.003 \cdot \left(\frac{273.15}{T}\right)^2\}$$

viscosity.dat

```
273.15   1.788e-3
283.15   1.307e-3
293.15   1.003e-3
303.15   0.799e-3
313.15   0.657e-3
323.15   0.548e-3
333.15   0.467e-3
343.15   0.405e-3
353.15   0.355e-3
363.15   0.316e-3
373.15   0.283e-3
```



Variations – 2 way coupling

Navier-Stokes

μ = const

Heat Transfer
+ convection

κ = const
c = const

Navier-Stokes

$\mu(T)$

Heat Transfer
+ convection

κ = const
c = const

Steady State Max Iterations = 1 → 50

Variations – 2 way coupling

- Copy the original solver input file (SIF)
- Open in editor of your choice (e.g., gedit)
  - apply the changes as suggested
  - change names of output files!
  - Include restart from earlier case:

```
Restart File = case.result
Restart Position = 0
```

  - The last line restarts from the last entry it found in **`case.result`**

Array 1

- Piecewise linear interpolation
- Alternative:

  `Real cubic`

  interpolates using cubic splines
- See SIF:

`coupled_array.sif`

```
Material 1
  Name = "Water (room temperature)"
  Viscosity = Variable Temperature
    Real
      273.15 1.788e-3 ! 0 Celsius
      283.15 1.307e-3
      293.15 1.003e-3
      303.15 0.799e-3
      313.15 0.657e-3
      323.15 0.548e-3
      333.15 0.467e-3
      343.15 0.405e-3
      353.15 0.355e-3
      363.15 0.316e-3
      373.15 0.283e-3 ! 100 Celsius
   End
```

Variations – 2 way coupling

- Save under case **`coupled_array.sif`**
- Run the case in serial:

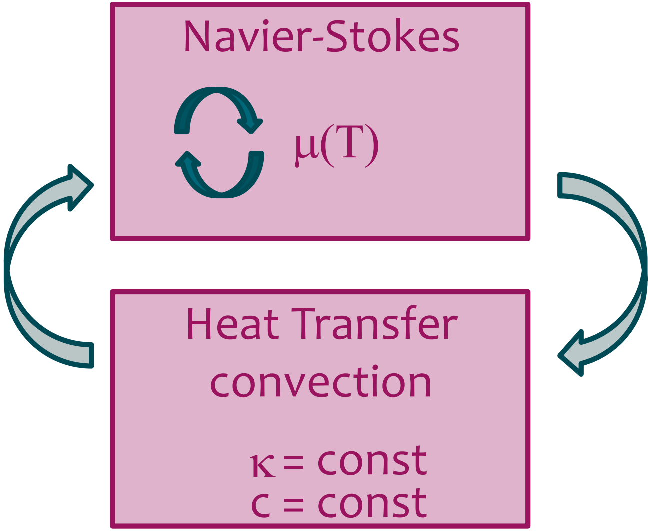
```
ElmerSolver coupled_array.sif > coupled_array.log &
```

  - Redirect output (good for checking performance)

Array 2

- Same as before, but now we switch to only one non-linear iteration for Navier-stokes
- Create new SIF:

`coupled_array_var.sif`



`Nonlinear System Max Iterations = 50 → 1`

MATC function

- Declare outside sections:

    Constant **mu0**

    Function **relativevisc**

- Call both using MATC from within **Material 1**

```
$ mu0 =  1.788e-3
$ function relativevisc(T){\
  a = -1.704;\
  b = -5.306;\
  c = 7.003;\
  z = 273.15/T;\
  _relativevisc = exp(a + b * z + c *(z^2));\
}
```

```
Material 1
  Name = "Water (room temperature)"
  Viscosity = Variable Temperature
    Real MATC "mu0 * relativevisc(tx)"
```

User Defined Function (UDF)

- Write a simple UDF in Fortran 90 that returns the value of viscosity from a given value of temperature
  viscosity1.f90
  - Pre-defined Header:

```
FUNCTION getWaterViscosity( Model, N, temperature ) &
RESULT(viscosity)
  USE DefUtils
  IMPLICIT NONE
  !------------ external variables --------------------------
-
  TYPE(Model_t) :: Model
  INTEGER :: N
  REAL(KIND=dp) :: temperature, viscosity
```

NB for F90: exponential function … exp()   multiplication …
*

User Defined Function (UDF)

- Compile it:

```
elmerf90 viscosity1.f90 -o viscosity1
```

- Re-write the Material 1 section:

```
Material 1
  Name = "Water (room
temperature)"
  Viscosity = Variable
Temperature
    Procedure "viscosity1"
"getWaterViscosity"
```

Alternative pre-processing tools for Elmer

ElmerTeam
CSC – IT Center for Science, Finland

CSC, 2018

Mesh generation capabilities of Elmer suite

- **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

Open CASCADE TECHNOLOGY

OpenFOAM

ParaView

vtk Visualization Toolkit

SALOME

Freefem++

deal.II

CODE SATURNE

Gmsh

Code_Aster

VisIt

FEniCS PROJECT

Netgen Mesh Generator
NETGEN is an automatic 3d tetrahedral mesh generator

Elmer

Trilinos

FreeCAD

PETSc

libMesh

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
  - Preliminary version for direct interface to Elmer
- **FreeCAD**
  - Open source community driven effort also based on OpenCascade
  - Preliminary version for direct interface to Elmer

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
- HyperMesh
  - Usable via the UNV export
- …

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

Mesh generation tools – Poll (5/2018)

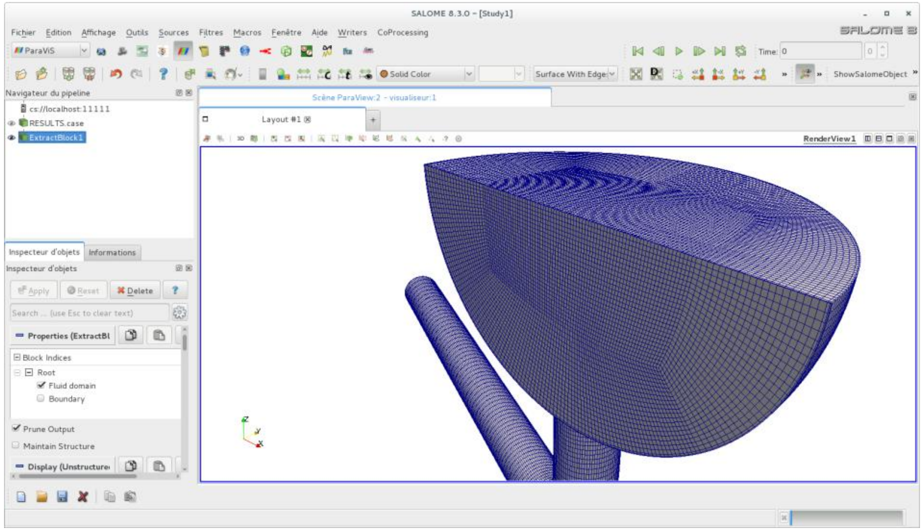
What mesh generation software do you use with Elmer?

ElmerGUI (netgen or tetgen plugins)	10	9%
Gmsh	49	44%
Netgen	11	10%
ElmerGrid (native .grd format)	9	8%
GiD	1	1%
Ansys	3	3%
Gambit	0	No votes
Comsol Multiphysics	1	1%
Salome	23	21%
Something else (please specify)	5	4%
	Total votes : 112	

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 third-party numerical codes.



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
- 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

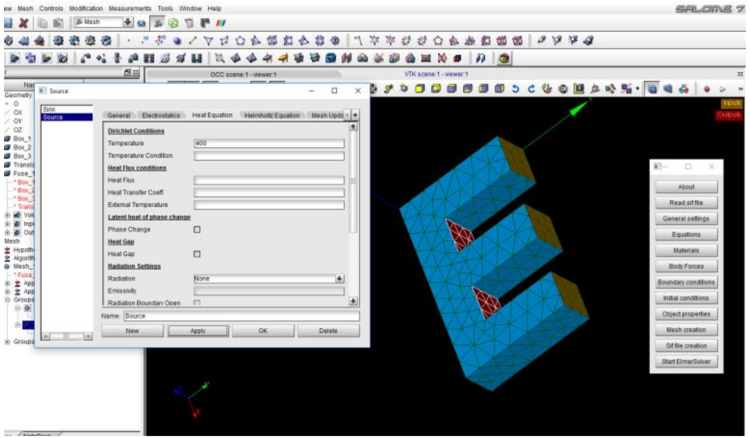
Elmer interface in SALOME
(Open source development by Rainer Jacob and Matthias Zenker)

Motivation:

- Salome already offers CAD, mesh creation and post processing via ParaView
- multiple loops of Salome <-> Elmer, if a geometry is not straight forward and mesh quality vs. time is critical

Solution

- Replace the ElmerGUI by an interface to the ElmerSolver that is directly accessible from Salome
- interface mimics the essential GUI functions for setting up and running a simulation
- Seamless integration into Salome by using the Object Browser and the object properties inside Salome
- Keep Elmers xml-solver files to reduce maintenance
- Log files for export/archiving the Solver output

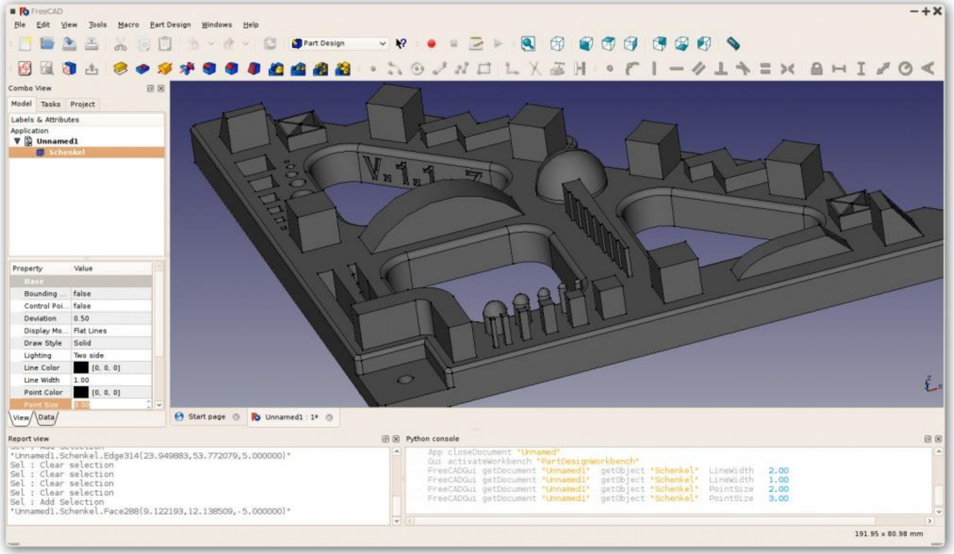


Demo: https://youtu.be/D2-dp4UxblY

FreeCAD

https://www.freecadweb.org/

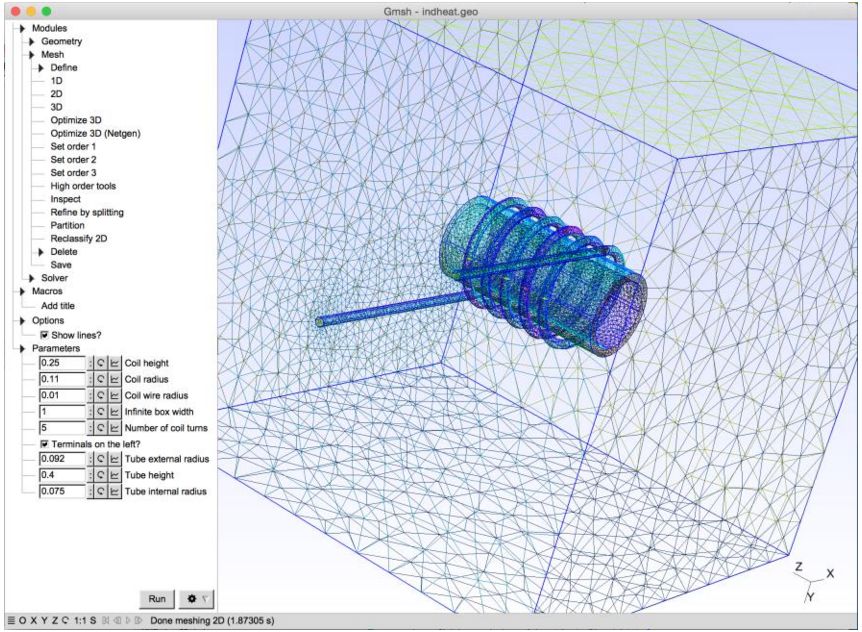
- FreeCAD is a parametric 3D modeler made primarily to design real-life objects of any size.
- Parametric modeling allows you to easily modify your design by going back into your model history and changing its parameters.
- FreeCAD is open-source and highly customizable, scriptable and extensible.
- FreeCAD is multiplatfom (Windows, Mac and Linux), and reads and writes many open file formats such as STEP, IGES, STL, SVG, DXF, OBJ, IFC, DAE and many others.



Gmsh

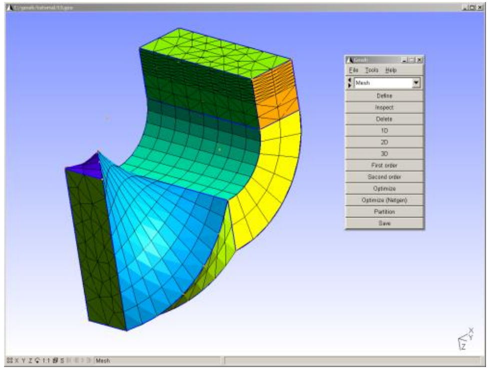
http://gmsh.info

- Written by C. Geuzaine and J.-F. 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

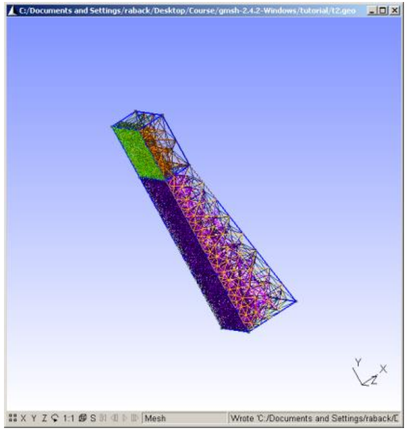
- Saving of the mesh in native gmsh format
  - Suffix .msh
- Usually saving all geometric entities is most robust method
  - Elmer automatically drops lower dimensional entities
  - Elmer renumbers BCs and bodies with 1,2,3,….



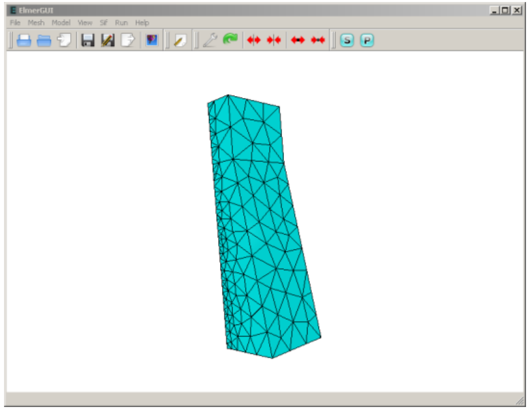
- In Gmsh:
  File -> Save as
    Filename: test.msh
      MSH Options
        Version 2.0 ASCII
        Save all (ignore physical groups)
- In ElmerGUI
  File -> Open : test.msh
- Or ElmerGrid:
  ElmerGrid 14 2 test.msh -autoclean
  (creates a mesh file in directory test)

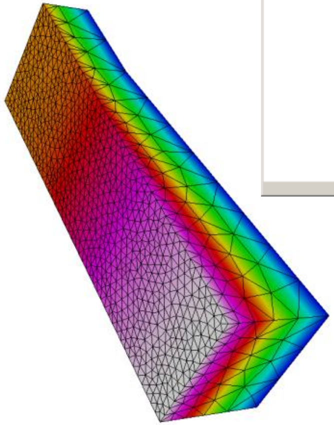
Exercise: Gmsh to Elmer export

- 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



Tutorial 2 of Gmsh

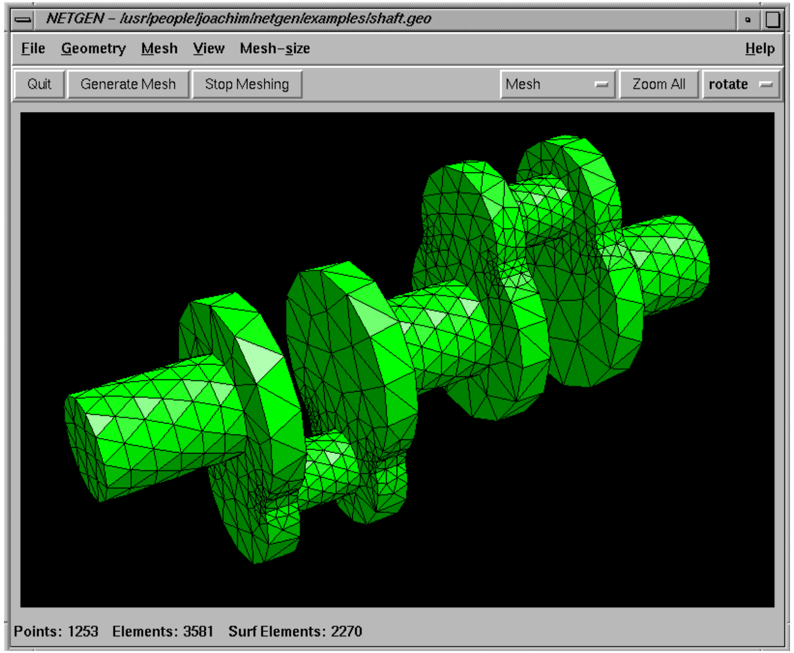




Netgen

http://ngsolve.org/

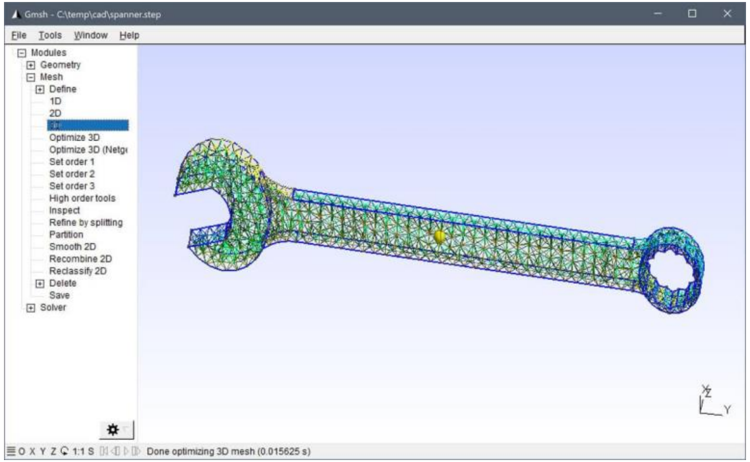
- Developed mainly by Joachim Schöberl
- An automatic 2D/3D tetrahedral mesh generator
- Accepts input from constructive solid geometry (CSG) or boundary representation (BRep) from STL file format
- Connection to OpenCASCADE deals with IGES and STEP files
- Modules for mesh optimization and mesh refinement
- LGPL library
- Netgen as a library is utilized by a large number of GUI projects
- Directly writes meshes in Elmer format (linear only)



GiD

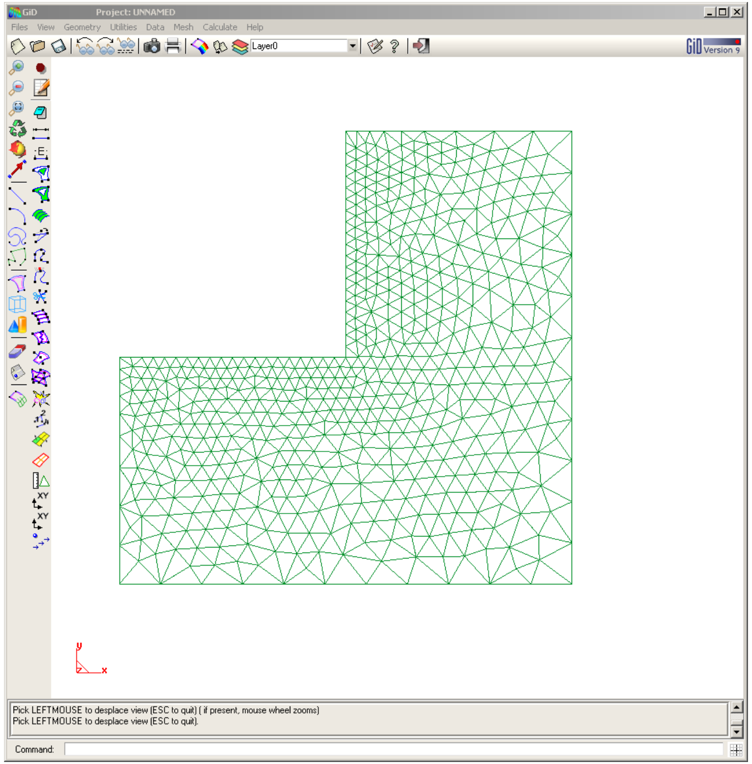
http://www.gidhome.com

- 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.nic.funet.fi/pub/sci/physics/elmer/macros/GiD2Elmer/



Summary of Pre-Processing Workflows in Elmer

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

- And many more....