

Elmer/Ice course

31st Oct, 1st & 2nd Nov 2016, Oslo

Introduction

Thomas ZWINGER (1)

Fabien GILLET-CHAULET(2)

Adrien GILBERT(3)

Olivier GAGLIARDINI (2)

(1) CSC - IT Center for Science Ltd. – Espoo - Finland

(2) LGGE - Grenoble - France

(3) University of Oslo - Oslo - Norway

Program

Monday 31st October

9:00-9:30 Arrival of the participants at University of Oslo

9:30-9:45 Welcome words by Thomas Schuler, general announcements

9:45-10:30 Introduction Elmer/ice (OG)

10:30-11:00 Coffee break

11:00-12:00 Toy flow-line model: basic diagnostic (TZ)

12:00 Lunch (sandwiches)

13:00-15:30 Toy flow-line model: thermo-mechanical coupling, sliding, prognostic runs (TZ)

15:30-16:00 Coffee break

16h00-17h30 Tête Rousse context and diagnostic (OG)

Program

Tuesday 1st November

9:00-10:30 Tête Rousse setup and diagnostic (OG)

10:30-11:00 Coffee break

11:00-12:00 Tête Rousse prognostic (OG)

12:00 Lunch (sandwiches)

13:00-14:30 Coupling ice flow and temperature (AG)

14:30-15:00 Coffee break

15:00-17:00 Coupling ice flow and temperature (AG)

19:00 - Course dinner (place to be specified)

Program

Monday 2nd November

9:00-10:30 Inverse modelling and SSA (FG)

10:30-11:00 Coffee break

11:00-12:00 Inverse modelling and SSA (FG)

12:00 Lunch (sandwiches)

13:00-14:30 Inverse modelling and SSA (FG)

14:30-16:00 Questions on your own modelling

16h00 End of the course

Short history of Elmer/Ice (not anymore so short...) 1/2

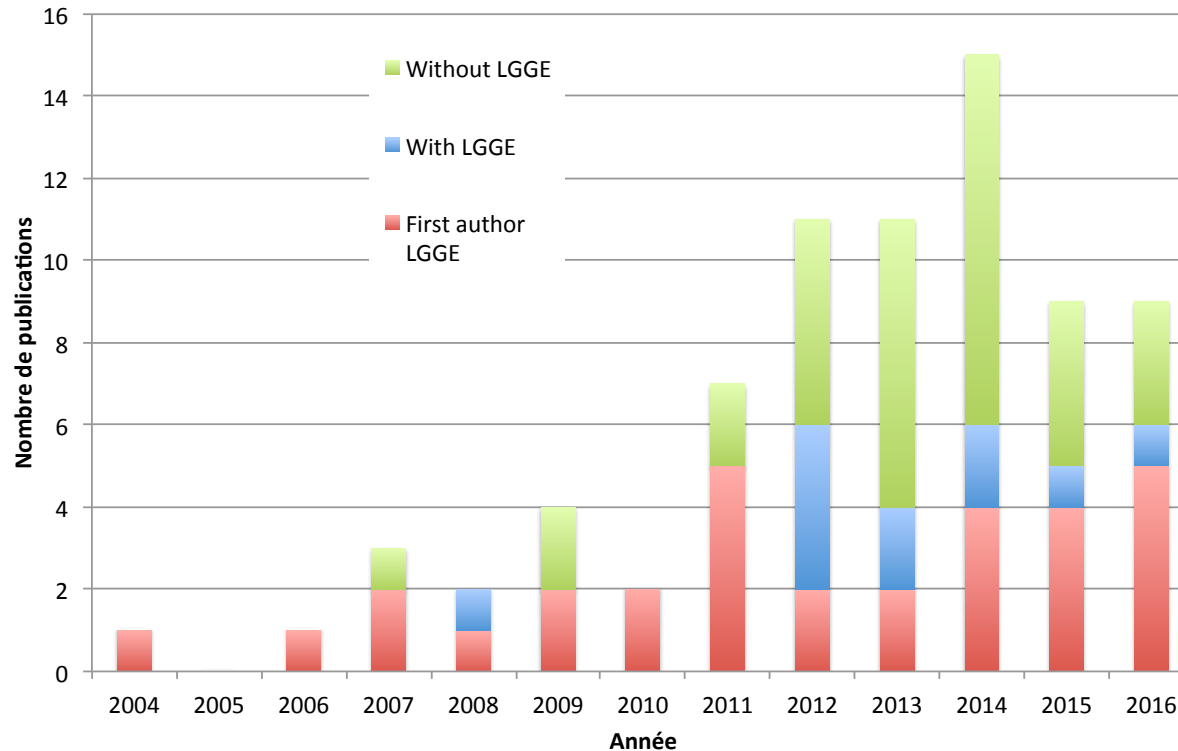
- ✓ EGU2002: OG was looking for a 3D FE code to model the flow of strain-induced anisotropic polar ice – meet TZ
- ✓ March 2003: OG visited CSC for few days: AIFlowSolver and FabricSolver partly implemented
- ✓ August 2005 – One year visit of OG at CSC (Anisotropy, cavity, glaciers, ISMIP tests, ...)
- ✓ February 2008 – First Elmer/Ice Course - Grenoble
- ✓ June 2011 – SVALI summer school – Finland
- ✓ 2012 – Elmer/Ice has now a website, a logo and a mailing list
- ✓ 2012 – Elmer/Ice comes as a Elmer Package – New wiki
- ✓ 2012 – Elmer/Ice course at UBC/SFU
- ✓ 2013 – Elmer/Ice courses at Univ. Washington and Univ. Alberta
- ✓ 9 April 2013 – First Elmer/Ice users meeting – EGU 2013

Short history of Elmer/Ice (not anymore so short...) 2/2

- ✓ May 2013 – Second SVALI summer school – Finland
- ✓ 2-day beginner Elmer/Ice course, 3-4 Oct. 2013, LGGE, Grenoble, France
- ✓ 3-day Elmer/Ice advanced workshop, 4-6 Nov. 2013, CSC, Espoo, Finland
- ✓ April 2014 – Second Elmer/Ice users meeting – EGU 2014
- ✓ 3-day beginner Elmer/Ice course, 27-29 Oct. 2014, IMO, Reykjavík, Iceland
- ✓ April 2015 – Third Elmer/Ice users meeting – EGU 2015
- ✓ 2-day beginner course, 1&2 Nov, CIC, Copenhagen, Denmark
- ✓ 3-day Elmer/Ice advanced workshop, 30 Nov, 1&2 Dec 2015, LGGE, Grenoble, France

and now Oslo...

A growing community



12 Courses ~ 220 participants

3 Elmer/Ice users meetings

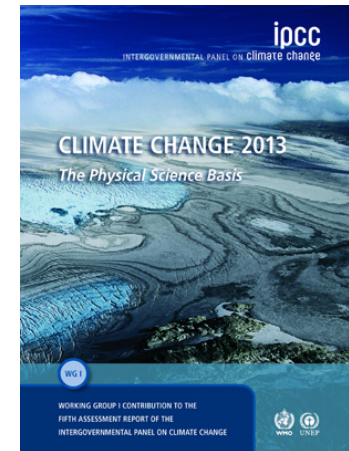
75 publications

Since few years, first authors are not only anymore only from CSC or LGGE...

Elmer/Ice applications

75 publications using Elmer/Ice since 2004

- ISMIP, MISMIP, MISMIP-3d
- 2D and 3D Grounding line dynamics
- Ice2sea and SeaRISE contributions (Greenland)
- Inverse methods (Variegated, Vestfonna ice-cap, GIS)
- Flow of anisotropic ice
- Glaciers
- 9 cited references including results from Elmer/Ice in the 5th IPCC report



see <http://elmerice.elmerfem.org/publications>

Elmer/Ice website

<http://elmerice.elmerfem.org/>

elmer/ice NEWS PUBLICATIONS CAPABILITIES USERS COMMUNITY FORUM COURSES TUTORIALS MATERIALS DOCUMENTATIONS

search...

Elmer/Ice New beginner Elmer/Ice course

Elmer/Ice

Open Source Finite Element Software for Ice Sheet, Glaciers and Ice Flow Modelling

Elmer/Ice is a full-Stokes, finite element, ice sheet / ice flow model. The aim of this website is to present the capabilities of Elmer/Ice and to distribute course materials and tutorials.

Elmer/Ice builds on Elmer, the open-source, parallel, finite element code, developed by the CSC-IT Center for Science Ltd. in Finland. Elmer/Ice is mainly developed by CSC-IT Center for Science Ltd., the LGGE and ILTS, but others contributors are welcome!

New beginner Elmer/Ice course

Written by [Olivier Gagliardini](#).

SVALI Elmer/Ice course, Reykjavik, Iceland, October 27-28, 2014

IMO in cooperation with CSC (Finland) and LGGE (France) will organize a 2-day Elmer/Ice course on the 27th and 28th of October 2014, just before the **Nordic Branch IGS meeting** in Iceland. The course is intended for persons who want to start using Elmer/Ice in their research projects with an option to go into special details on the second day. The course is sponsored by the Nordic Centre of Excellence, SVALI. There are in maximum 18 seats which by preference will be offered to SVALI members. The rest of the seats are given on a first-come-first-get principle. The course itself is free of charge. Participants are responsible for their own accommodation and travel (see instructions below). Further it is expected, that participants bring along their own laptop with a working Elmer/Ice environment installed (instructions will follow later).

Lectures are in English.
Lecturers: Olivier Gagliardini, Thomas Zwinger.
Venue: IMO, Bústaðavegur 9, Reykjavik.
Travel: For air travel to Iceland see "icelandair.com" and "dohop.com" (further instructions about how to get to the IMO will follow later).
Accommodation: For reasonably priced guesthouses check "kexhostel.is/" and "guesthousereykjavik.com/".
Registration: By email to Anna Sinisalo (a.k.sinisalo[at]geo[dot]julo[dot]no), latest by 11th October, 2014.

[Tweet](#)

Latest News

- [A thermal regime model for polythermal glaciers](#)
- [New beginner Elmer/Ice course](#)
- [How old is the ice beneath Dome A, Antarctica?](#)
- [Elmer/Ice at IGS Chamoinx 2014](#)
- [Environmental influences on the calving behaviour of Halheim](#)

Tweets

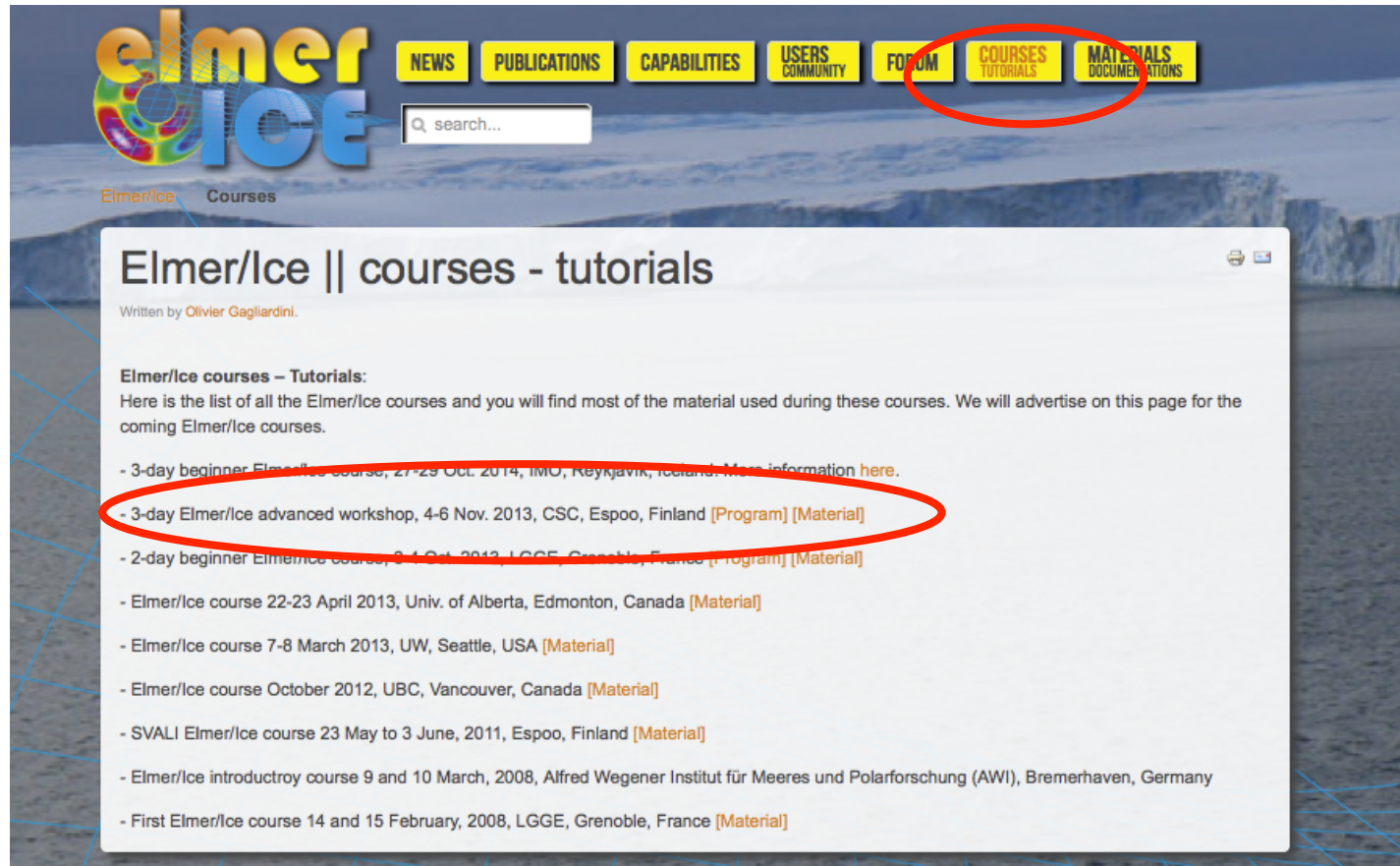
[Follow](#)

Elmer/Ice @ElmerIce1 26 Aug
New beginner Elmer/Ice course
elmerice.elmerfem.org/45-new-beginne...
Expand

Elmer/Ice @ElmerIce1 21 Aug
A new beginner Elmer/Ice ice course in Iceland, end of October. Book your seat!
elmerice.elmerfem.org/45-new-beginne...

Elmer/Ice website

<http://elmerice.elmerfem.org/>



Much more material available than what we will present today

Elmer/Ice wiki <http://elmerice.elmerfem.org/wiki/doku.php>

[[start]]

ELMER/ICE WIKI

Trace: • start

Show pagesource Recent changes Sitemap Login

Search

- Home
- Problems
- Solvers
- User Functions
- Meshing Tools
- Tips and Tricks
- Meetings and Courses
- Compilation of Elmer/Ice
- Links

Welcome to the Elmer/Ice wiki

Elmer is an open-source, parallel, Finite Element code, mainly developed by the CSC-IT Center for Science Ltd. in Finland. Elmer/Ice builds on Elmer and includes developments related to glaciological problems. On this page you will find documentation and examples of the various solvers and user functions developed for **glaciological** applications using Elmer/Ice. Building Elmer/Ice on top of an existing Elmer installation is explained in the [Compilation Section](#).

The [Problems Section](#) presents the various categories of glaciological problems that can be solved using Elmer/Ice.

The [Solvers Section](#) and the [User Functions Section](#) describe the glaciology related solvers and user functions, respectively, that can be used to solve these problems.

Tools that can be used to mesh glacier and ice-sheet geometry are presented in the [Meshing Section](#).

The [Tips and Tricks Section](#) gives some useful demo of MATC, Post-treatments of results and more.

The [Courses Material Section](#) contains presentation as well as material proposed in the framework of the Elmer/Ice courses dispensed since 2008.

Some useful links are given in the [Links Section](#).

Scientific publications presenting glaciological applications with Elmer/Ice are listed in the [Elmer/Ice website](#).

start.txt · Last modified: 2012/12/03 17:45 by tzwinger

Elmer/Ice mailing list

To subscribe to the Elmer/Ice list elmerice@elmerfem.org, just sent an email to majordomo@elmerfem.org, with in the body the text:

subscribe elmerice

If you do not know how to use mailing lists run by majordomo you may sent a mail with "help" in the message body.

Elmer/Ice Forum

Under

<http://www.elmerfem.org> :

•Go to **Elmer Forum**:
find answers on all
aspects of Elmer

•Click on **Elmer/Ice**
link: find answers
specific to Elmer/Ice

•To get access:
Register in upper right
corner

Elmer Discussion Forum • Index page - Mozilla Firefox

www.elmerfem.org/forum/index.php

phpBB Elmer Discussion Forum
Bulletin Board for Elmer FEM Users

Search... Search
Advanced search

Board index

FAQ Register Login

It is currently 02 Sep 2013, 10:54

View unanswered posts • View active topics

FORUMS	TOPICS	POSTS	LAST POST
General General discussion about Elmer	345	1289	by GastónGarcía 29 Aug 2013, 22:01
Installation & compilation Discussion about building and installing Elmer	166	680	by Jyh-Shyong 30 Aug 2013, 18:02
ElmerSolver Numerical methods and mathematical models of Elmer	944	4108	by drueffer 30 Aug 2013, 16:54
ElmerGUI The graphical user interface of Elmer	228	908	by raback 26 Aug 2013, 00:05
ElmerPost Post processing utility for Elmer	90	346	by Edmund 30 Jul 2013, 11:01
Elmer/Ice Extension of Elmer in computational glaciology	8	27	by tzwinger 22 Aug 2013, 13:56
External tools Mesh generators, CAD programs, and other tools	113	558	by NickR7 30 Aug 2013, 18:04
Software development Discussion about coding and new developments	37	120	by Takala 22 Aug 2013, 09:02
Bug reports Clearly defined bug reports and their fixes	80	230	by millm 29 Aug 2013, 12:55
Contributed Cases Elmer cases by the users for the users	15	33	by sebastien ROUQUETTE 29 Apr 2013, 14:49
HPC High Performance Computing with Elmer	3	5	by madtom1999 21 Oct 2012, 15:34
Commerical services A forum for commercial service requests and offerings	3	3	by aether 12 Dec 2012, 11:33

ANNOUNCEMENTS	TOPICS	POSTS	LAST POST
Updates Updates in software, documentation, sites etc.	20	83	by mzenker 08 Jul 2013, 16:20
Events Courses, user meetings, seminars etc.	14	17	by raback 12 Apr 2013, 13:54

MISCELLANEOUS	TOPICS	POSTS	LAST POST
Testing Here you can test posting, attachments, ...	4	6	by Takala 03 Apr 2013, 10:16

Elmer/Ice on Twitter @ElmerIce1

elmer/ice
@ElmerIce1

Open Source Finite Element Software for Ice Sheet, Glaciers and Ice Flow Modelling

elmerice.elmerfem.org

TWEETS 24 ABONNEMENTS 19 ABONNÉS 69

Tweets Tweets & réponses

Elmer/Ice @ElmerIce1 · 26 août
Two new Elmer/Ice courses this fall, for beginner and confirmed. Don't miss the date and register now! elmerice.elmerfem.org/63-two-elmer-i

Elmer/Ice @ElmerIce1 · 18 août
Steep cold hanging glaciers located below 4000m are likely to become partially temperate by the end of this century elmerice.elmerfem.org/62-thermal-cha...

Important links (summary)

Elmer at CSC (documentation, how to install, ...)

<http://www.elmerfem.org/>

<http://www.csc.fi/english/pages/elmer>

Elmer Forum

<http://elmerfem.org/forum/>

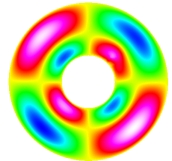
Elmer/Ice webpage

<http://elmerice.elmerfem.org/>

Elmer/Ice wiki

<http://elmerice.elmerfem.org/wiki/doku.php?id=start>

Elmer/Ice in relation to Elmer



Elmer is an open-source, parallel, Finite Element code, mainly developed by the CSC-IT Center for Science Ltd. in Finland.

Elmer is constantly developed towards improved performance, utilizing international projects such as FP7 PRACE and HPC Europa2.



Elmer/Ice builds on Elmer and includes developments related to glaciological problems. Elmer/Ice includes a variety of dedicated solvers and user functions for glaciological applications and its development is supported by various groups and funding...



norden

Top-level Research Initiative



Elmer/Ice Package

All the Solvers, User Functions and Meshers presented on the Elmer/Ice wiki comes as an Elmer/Ice package on the Elmer distribution (in `elmerice/`)

To get Elmer/Ice installed, add the following option to the cmake build command:

```
-DWITH_ElmerIce:BOOL=TRUE
```

To use it (in the SIF file):

```
Procedure = File "ElmerIceSolvers" "NameSolver"
```

or

```
Procedure = File "ElmerIceUSF" "NameUSF"
```

Important notices

In this course

- We will not teach finite element method (can give references)
- We will focus on some technical aspects of using Elmer for glaciological applications

What we expect from this course ?

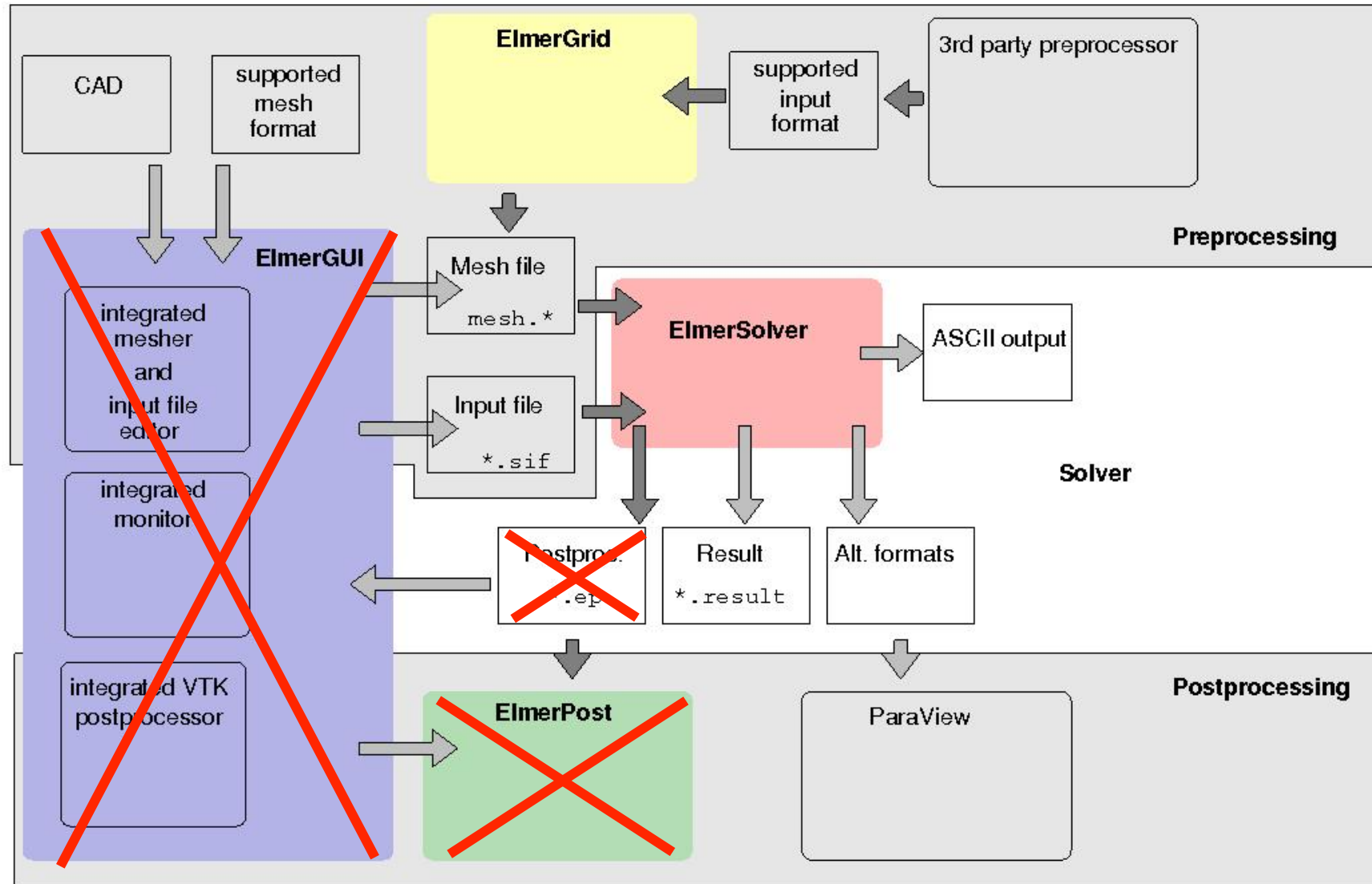
- giving you a kick-start in Elmer/Ice
- some fruitful collaborations to begin

Elmer/Ice capabilities

- **Full-Stokes** equations but also SIA, SSA, diagnostic or transient
- Various **rheologies** (Glen's law, firn/snow and anisotropic flow laws)
- **Temperature** solver accounting for the upper limit at melting point
- **Transport equations** for density, fabric, age ...
- **Post-processing solver** for strain-rate and stress fields
- Various **friction laws** (Weertman, effective-pressure dependent friction law)
- **Free surface evolution** as a contact problem (Grounding line dynamics)
- **Inverse methods** (linear adjoint and Arthern and Gudmundsson 2010 methods)
- Tools or plug-ins for **meshing** (YAMS, external and internal extrusion of footprint)
- **Highly parallel** Stokes solver
- **Basal hydrology** (2 approaches, one in the distribution)
- **Calving** (3 approaches, not yet in the distribution)
- **Damage mechanics**

How does it work ?

Elmer structure



Sequence of a serial simulation

- build a mesh in Elmer format, i.e. a directory containing
`mesh.header`, `mesh.nodes`, `mesh.element`, `mesh.boundary`
- fill in a solver input file (`mysif.sif`)
- compile object files linked with Elmer of your user functions and solvers (if needed)
- Execute :
\$ **ElmerSolver mysif.sif**
- Should create a *.vtu files (output files in vtu format)
- Visualise :
\$ **paraview**

We will see

- how to construct a simple mesh
- what is the content of a sif file
- how to execute
- how to visualise the results

How to get a mesh ?

Different possibilities to get a mesh

- use **ElmerGrid** alone
- use **another mesher** (gmsch, gambit) and then transform it in Elmer format - ElmerGrid can do this for many other mesh formats (just launch ElmerGrid without any argument to get list)
- Glacier particularities :
 - Small aspect ratio (horizontally elongated elements)
 - In 3D, mesh a footprint with an unstructured mesh, and then vertically extrude it (externally or internally)

will see this later during the course...

ElmerGrid

- command line tool for mesh generation
- native mesh format: `.grd`
- help : just execute : `$ ElmerGrid`
- possible to import meshes produced by other free or commercial mesh generators (Ansys, Abaqus, Fluent/Neutral, Comsol, gmsh, ...)
- Examples :

```
$ ElmerGrid 1 2 my_mesh.grd  
$ ElmerGrid 14 2 my_gmsh_mesh.msh -autoclean  
$ ElmerGrid 14 5 my_gmsh_mesh.msh -autoclean
```

Solver Input File (sif)

Example of sif file

- Comments start with !
- Not case sensitive
- Avoid non-printable characters (e.g., tabulators for indents)
- A section always ends with the keyword `End` or use `::`
- Parameters not in the Keyword DB need to be casted by types:
Integer, Real, Logical, String and File
- `Parametername (n, m)` indicates a $n \times m$ array

- Sections are

```
Header
Constants
Simulation
Solver i
Body i
Equation i
Body Force i
Material i
Initial Condition i
Boundary Condition i
```

```
Body Force 1
Heat Source = 1.0
End

OR

Body Force 1 :: Heat Source = 1.0
```

Example of sif file

```
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
!!
!! Elmer/Ice Course - Application Step0 !!
!!
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
! Updated May 2011

check keywords warn
echo on

Header
  Mesh DB "." "square"
End

Constants
! No constant needed
End

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
Simulation
  Coordinate System = Cartesian 2D
  Simulation Type = Steady State

  Steady State Min Iterations = 1
  Steady State Max Iterations = 1

  Output File = "ismip_step0.result"
  Post File = "ismip_step0.vtu"
  max output level = 100
End

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
Body 1
  Equation = 1
  Body Force = 1
  Material = 1
  Initial Condition = 1
End

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
Initial Condition 1
  Pressure = Real 0.0
  Velocity 1 = Real 0.0
  Velocity 2 = Real 0.0
End

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
Body Force 1
  Flow BodyForce 1 = Real 0.0
  Flow BodyForce 2 = Real -1.0
End
```

- **Header** declares where to search for the mesh
- If any **constants** needed (i.e. Gas constant)
- **Simulation**
 - Type of coordinate system
 - Steady or Transient
 - If transient: time stepping parameters
 - Output files (to restart a run) and VTU file
 - Output level : how verbose is the code?
 - Restart information (optional)
- In **Body** are assigned the Equation, Body Force, Material and Initial Condition
- In **Initial Condition** sets initial variable values
- In **Body Force** specify the body force entering the right side of the solved equation

Example of sif file

```
!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
Material 1
  Density = Real 1.0

  Viscosity Model = String "power law"
  Viscosity = Real 1.0
  Viscosity Exponent = Real 0.33333333333333333333
  Critical Shear Rate = Real 1.0e-10
End

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
Solver 1
  Equation = "Navier-Stokes"

  Stabilization Method = String Bubbles
  Flow Model = String Stokes

  Linear System Solver = Direct
  Linear System Direct Method = umfpack

  Nonlinear System Max Iterations = 100
  Nonlinear System Convergence Tolerance = 1.0e-5
  Nonlinear System Newton After Iterations = 5
  Nonlinear System Newton After Tolerance = 1.0e-02
  Nonlinear System Relaxation Factor = 1.00

  Steady State Convergence Tolerance = Real 1.0e-3
End

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!

Equation 1
  Active Solvers(1)= 1
End

!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!!
Boundary Condition 1
  Target Boundaries = 1
  Velocity 2 = Real 0.0e0
End

Boundary Condition 2
  Target Boundaries = 4
  Velocity 1 = Real 0.0e0
End

Boundary Condition 3
  Target Coordinates(1,2) = Real 0.0 1.0
  Target Coordinates Eps = Real 1.0e-3
  Pressure = Real 0.0e0
End
```

- In **Material** sets material properties for the body (can be scalars or tensors, and can be given as dependent functions)
- In **Solver** specifies the numerical treatment for these equations (methods, criteria of convergence,...)
- In **Equation** sets the active solvers
- **Boundary Condition**
 - Dirichlet: Variablename = Value
 - Neumann: special keyword depending on the solver
 - Values can be given as function

Variable defined as a function

1) Tables can be use to define a piecewise linear (cubic) dependency of a variable

```
Density = Variable Temperature
```

```
Real cubic
```

```
0 900
```

```
273 1000
```

```
300 1020
```

```
400 1000
```

```
End
```

Outside range: Extrapolation!

2) MATC: a library for online (in SIF) numerical evaluation of mathematical functions

```
Density = Variable Temperature
```

```
MATC "1000*(1 - 1.0e-4*(tx-273.0))"
```

or as constant expressions

```
Viscosity Exponent = Real $1.0/3.0
```

Evaluated every time

Evaluated once

3) Build your own user function

```
Density = Variable Temperature
```

```
Procedure "filename" "proc"
```

filename should contain a shareable (.so on Unix) code for the user function
whose name is `proc`

Example of User Function

```
FUNCTION proc( Model, n, T ) RESULT(dens)
USE DefUtils
IMPLICIT None
TYPE(Model_t) :: Model
INTEGER :: n
REAL(KIND=dp) :: T, dens

    dens = 1000*(1-1.0d-4 *(T-273.0_dp))

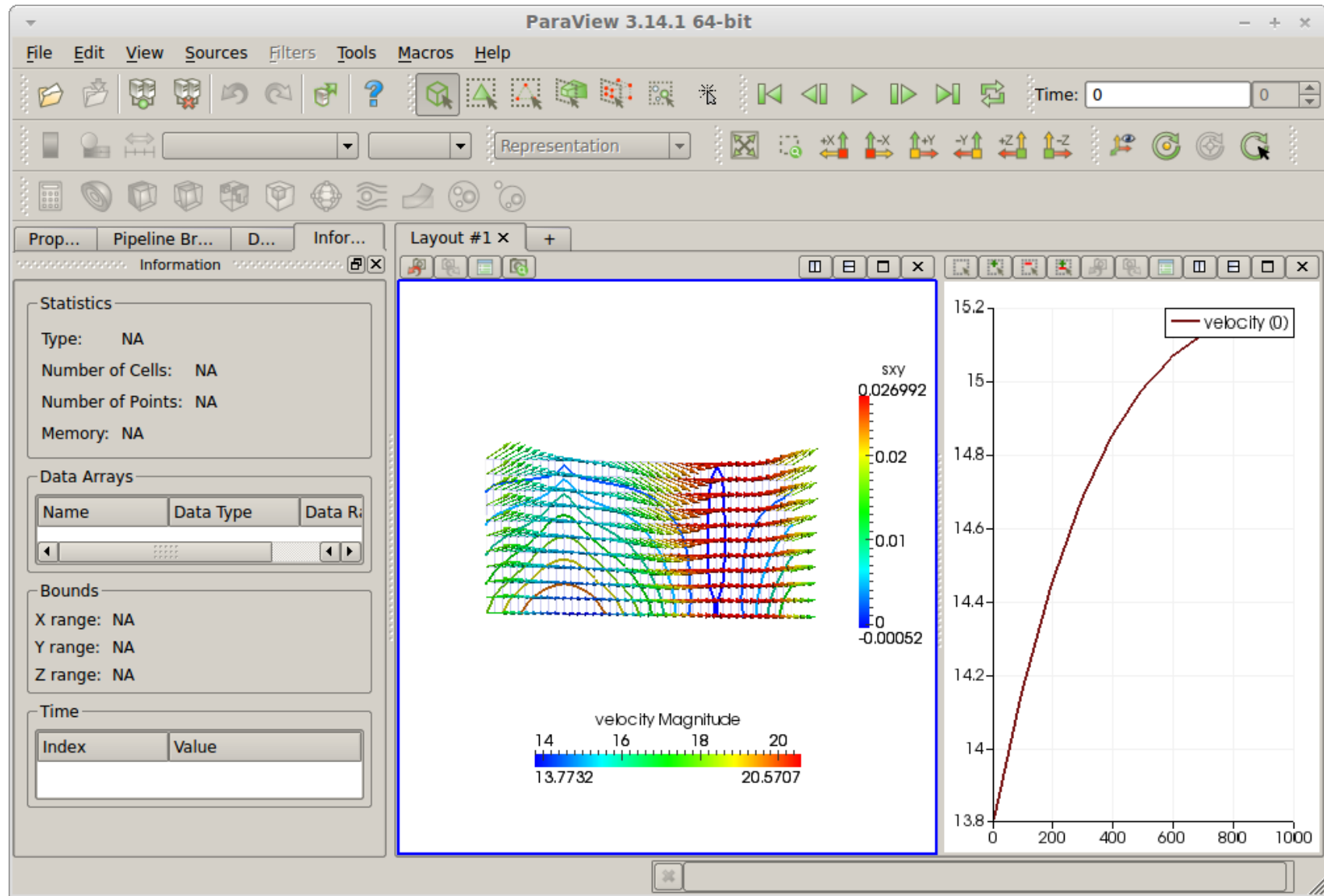
END FUNCTION proc
```

Compilation tools: `elmerf90`

```
$ elmerf90 filename.f90 -o filename.so
```

How to visualise results

Paraview



ASCII Based Output

- SaveScalars e.g. CPU time, mean, max, min of a variable, Flux
- SaveLine save a variable along a line (boundary or a given line)
- SaveMaterials save a material parameter like a variable

Example:

```
Solver 3
  Exec Solver = After All
  Procedure = File "SaveData" "SaveLine"
  Filename = "ismip_surface.dat"
  File Append = Logical False
End

Solver 4
  Exec Solver = After TimeStep ! For transient simulation
  Procedure = File "./MySaveData" "SaveScalars"
  Filename = "ismip_scalars.dat"
  File Append = Logical True ! For transient simulation

  Variable 1 = String "Flow solution"
  Operator 1 = String "Volume"

  Variable 2 = String "velocity 1"
  Operator 2 = String "Max Abs"

  Variable 3 = String "Flow solution"
  Operator 3 = String "Convective flux"

  Variable 4 = String "cpu time"

  Variable 5 = String "cpu memory"
End
```

```
! Upper Surface
Boundary Condition 3
  Target Boundaries = 3
  Save Line = Logical True
  Flux integrate = Logical True
End
```