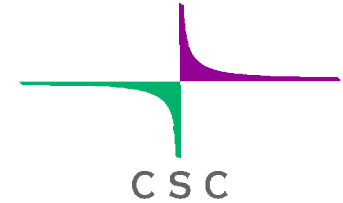


Laboratoire de Glaciologie et Géophysique de l'Environnement



First Elmer/Ice course

14-15 February 2008

Thomas ZWINGER ⁽¹⁾ and Olivier GAGLIARDINI ⁽²⁾

adapted from a presentation given by Peter Rabåck (CSC)
during the First Elmer User meeting (2006)

How to get a mesh?

- (1) CSC-Scientific Computing Ltd., Espoo - Finland
- (2) LGGE - Grenoble - France

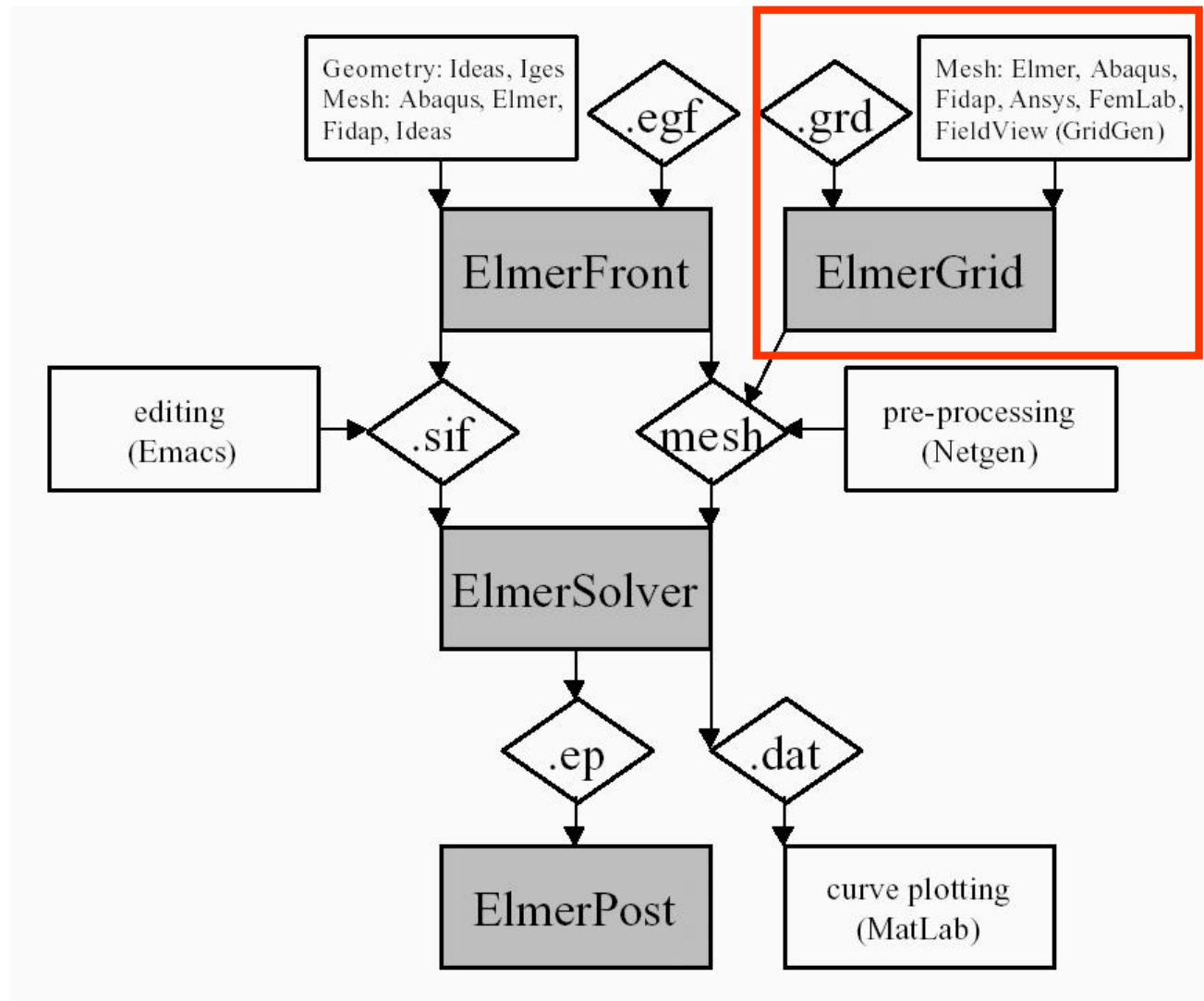
How to get a mesh ?

- ElmerGrid: the Elmer mesher
 - capability overview
 - importing mesh from other mesher
- Specific problems for glacier mesh
 - reserve of elements for growing glaciers
(theoretical problem, it does not exist anymore ...)
 - mesh coincide with the bed and surface data grid
(case of synthetic data)
 - not structured mesh in the horizontal plane (bed and surface grid do not coincide with the mesh)
 - Mesh from Digital Elevation Model
- Example: Different methods to mesh ISMIP-HOM B tests

ElmerGrid basis

- command line tool for mesh generation
- executable name: `ElmerGrid`
- native mesh format: `*.grd`
- native command format: `*.eg`
- short help when no parameters given
- Input format for ElmerSolver (output of ElmerGrid):
 - `mesh/mesh.header`
 - `mesh/mesh.nodes`
 - `mesh/mesh.elements`
 - `mesh/mesh.boundary`

Workflow in Elmer simulations



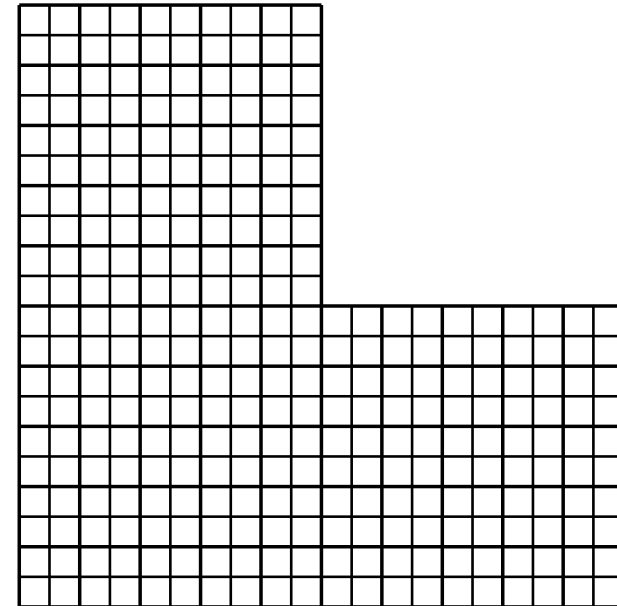
Mesh generation 2D

- mesh is thought to consist of subcells with a number
- the subcell grid is given dimensions and relative (or absolute) mesh density parameters
- boundary conditions are determined by neighbourhood relations
- desired number of elements is distributed considering the given constraints
- element order and type may be defined
- subcells boundaries may be mapped to non-rectangular forms
- same mesh may be used to create a number of meshes with different number of elements, or different domains, etc.

Mesh generation 2D - Example

ElmerGrid file for a L-Shaped mesh (angle.grd)

```
Version = 210903
Coordinate System = Cartesian 2D
Subcell Divisions in 2D = 2 2
Subcell Sizes 1 = 1 1
Subcell Sizes 2 = 1 1
Material Structure in 2D
1 0
1 1
End
Materials Interval = 1 1
Boundary Definitions
! type out int double of the boundaries
1 0 1 1
End
Element Degree = 1
Surface Elements = 300
```



Two ways to generate the mesh

```
> ElmerGrid 1 2 angle.grd
```

Or a angle.eg file

Input File = angle

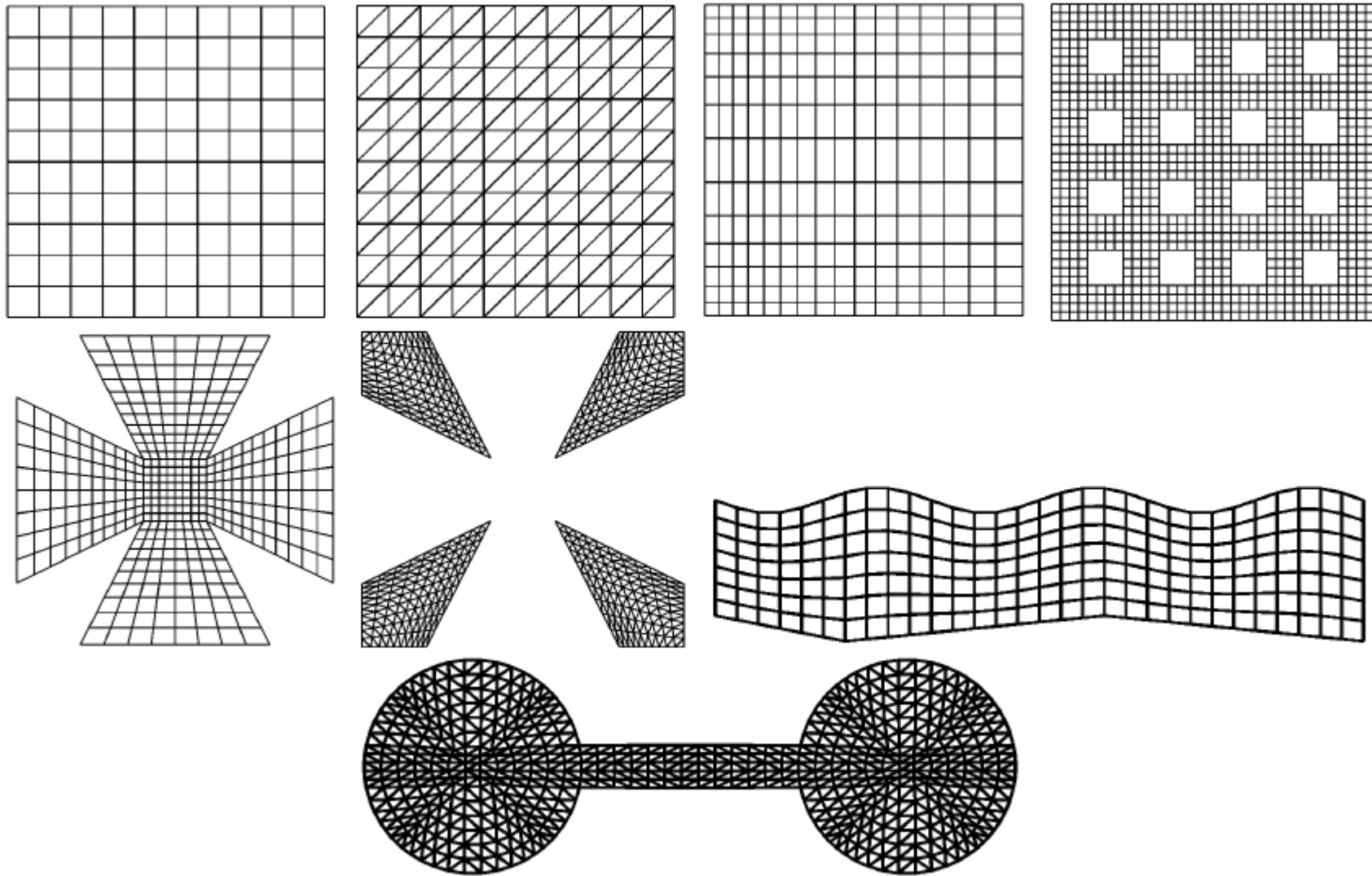
Output File = angle

Input Mode = ElmerGrid

Output Mode = ElmerSolver

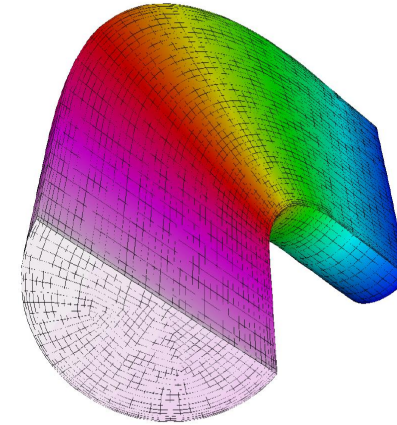
```
> ElmerGrid angle.eg
```

Other examples

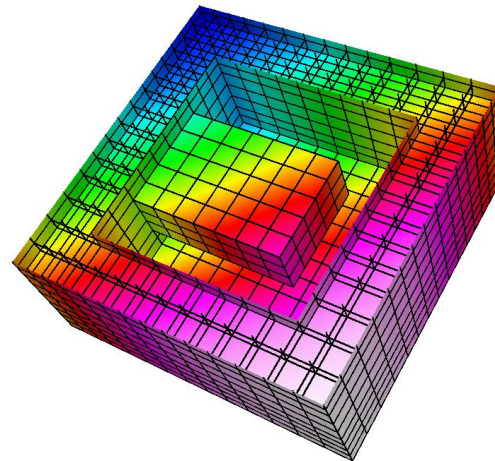


Mesh generation 3D

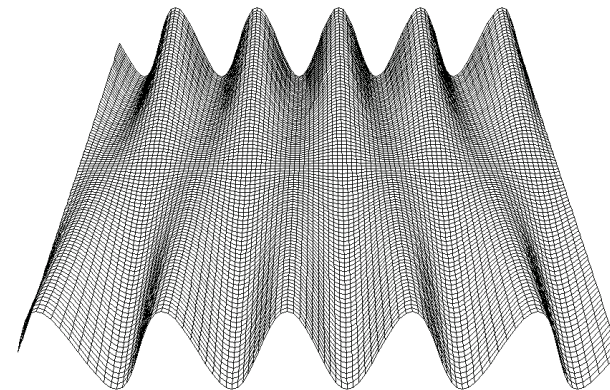
- Rotation of 2D meshes



- Extrusion of 2D meshes

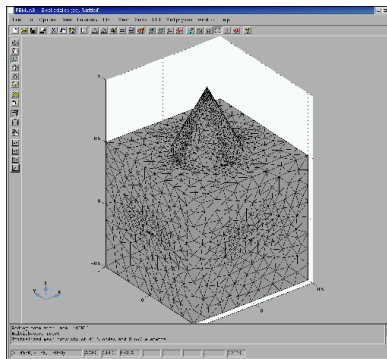


- Mapping of 2D meshes into third dimension

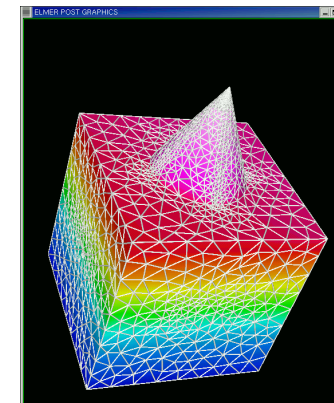


Mesh importation

- Important way to use ElmerGrid in mesh generation is to import meshes produced by other free or commercial mesh generators.
- Each import format has a corresponding command line
- Parsers for the mesh generators have been written case-by-case and therefore no general applicability is to be expected
- Supported formats include: Ansys, Abaqus, Fidap (Gambit), Comsol Multiphysics, FieldView (Gridgen), Medit, and Gmsh.



e.g.: Comsol multiphysics
to Elmer

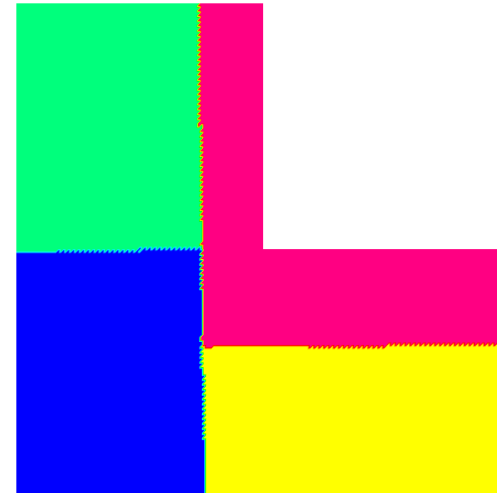


Mesh partitioning

- Mesh partitioning using Metis or regular partitions



Input File = angle
Output File = angle
Input Mode = ElmerGrid
Output Mode = ElmerPost
Metis = 4



Input File = angle
Output File = angle
Input Mode = ElmerGrid
Output Mode = ElmerPost
Partition = 2 2

How to get a mesh ?

- ElmerGrid: the Elmer mesher
 - capability overview
 - importing mesh from other mesher
- Specific problems for glacier mesh
 - reserve of elements for growing glaciers
 - mesh coincide with the bed and surface data grid (case of synthetic data)
 - not structured mesh in the horizontal plane (bed and surface grid do not coincide with the mesh)
 - Mesh from Digital Elevation Model
- Example: Different methods to mesh ISMIP-HOM B tests

Mesh grid = data grid

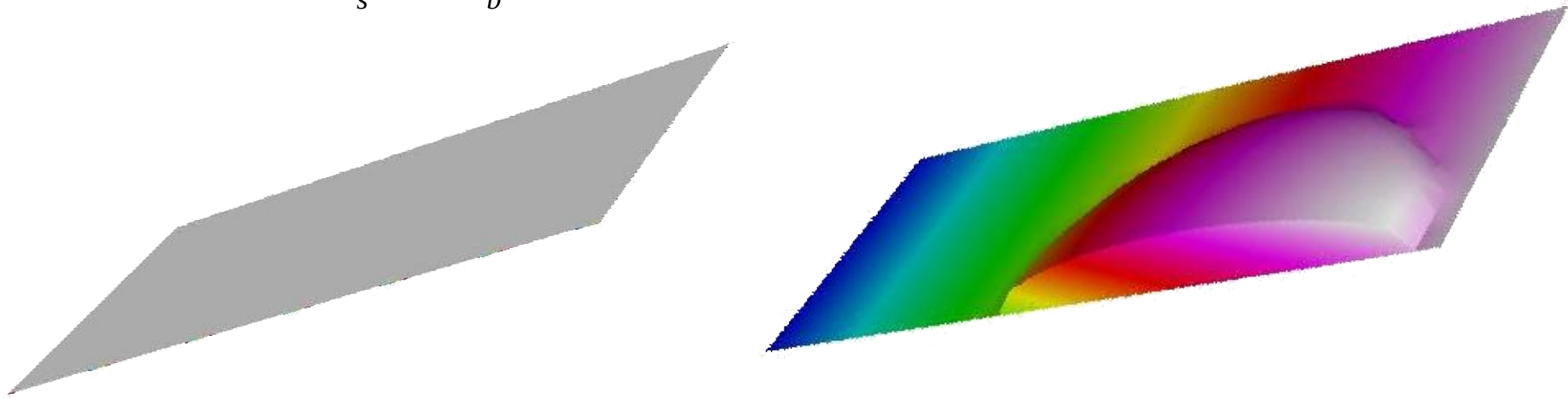
This is the case for a regular mesh in the horizontal plane and/or for data given analytically.

Method (for 3D):

- make a box $L_x * L_y * 1.0$ with ElmerGrid
- Deform vertically the box as

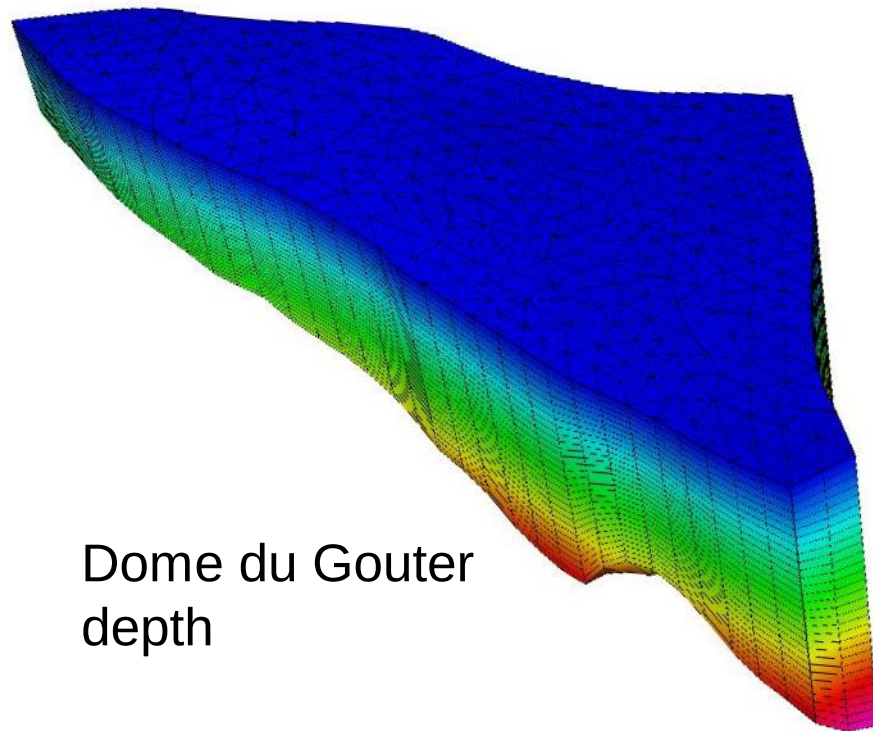
$$z \rightarrow z_b + (z_s - z_b) \times z$$

where z_s and z_b are the surface and bedrock elevations

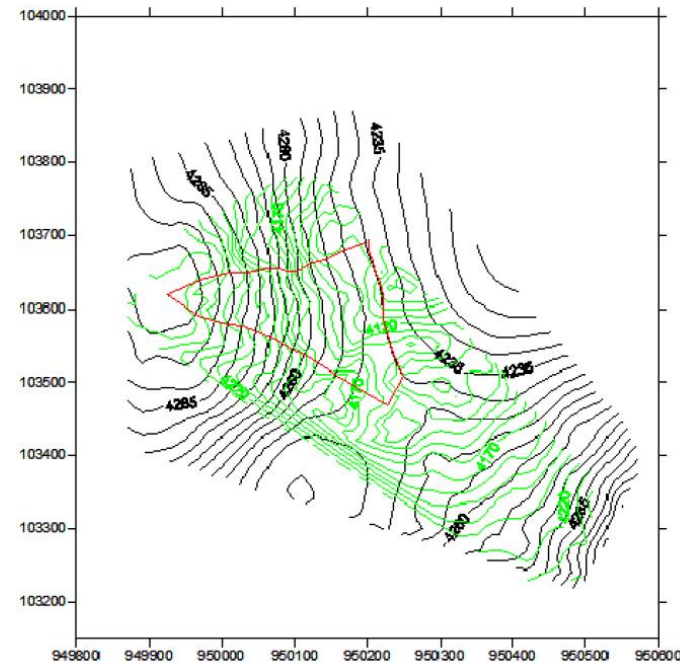


Mesh grid $\neq$ Data grid

Must interpolate from the grid data the nodal values of bedrock and surface elevation, but also accumulation, surface temperature, ...

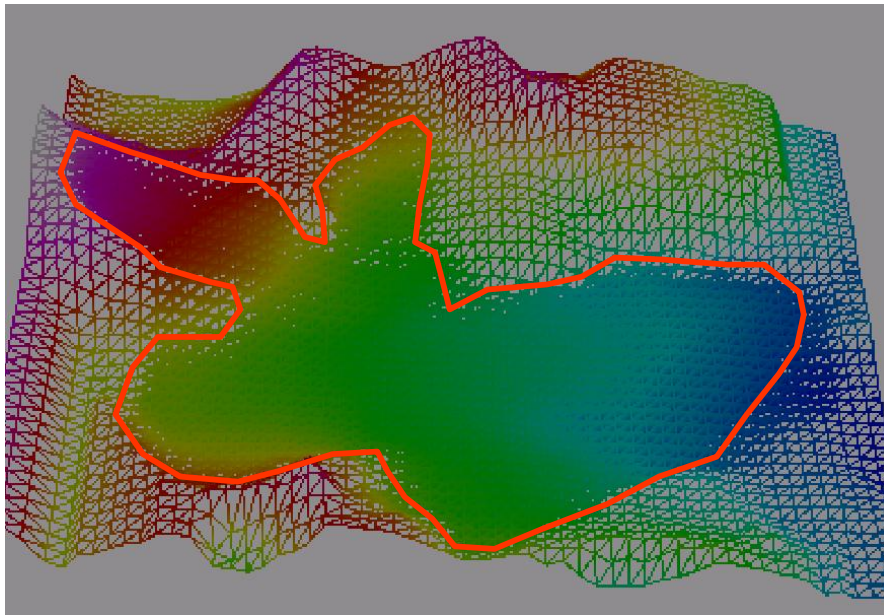


Dome du Gouter
depth



Mesh strategy for glacier

In the case of evolving geometry, a larger area than the glacier itself has to be meshed.

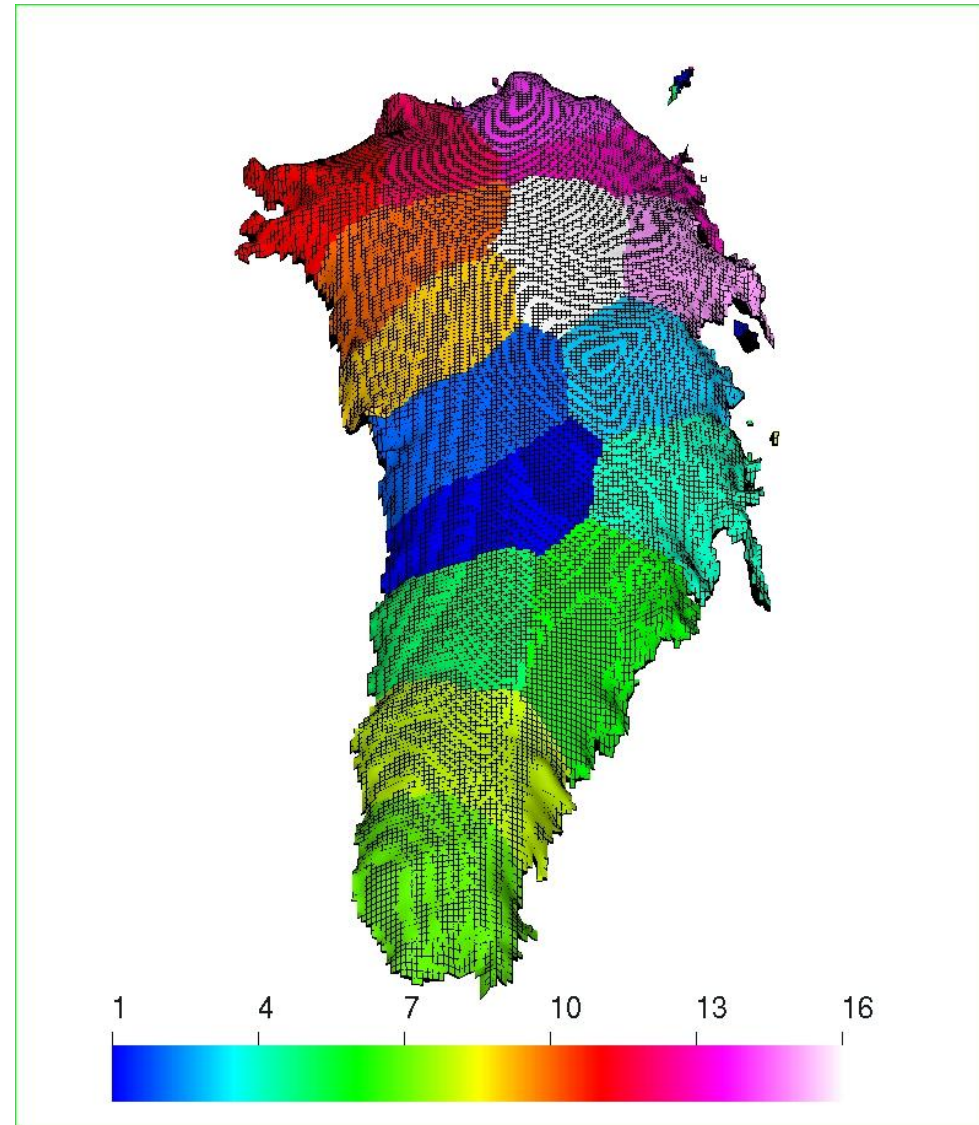


Saint Sorlin glacier

The 'waiting' elements must have a non-zero elevation

Mesh from DEM

- **ElmerDEM.c**
- Extruded mesh from DEM
- Saves into serial mesh database
- Partitioned later with **ElmerGrid 2.2**
meshDB -metis 16 4
- Code not part of Elmer distro
- Ask Thomas Zwinger for source
- Picture shows GIS with a horizontal resolution of $10 \times 10 \text{ km}^2$ and 12 horizontal layers (300k elements)

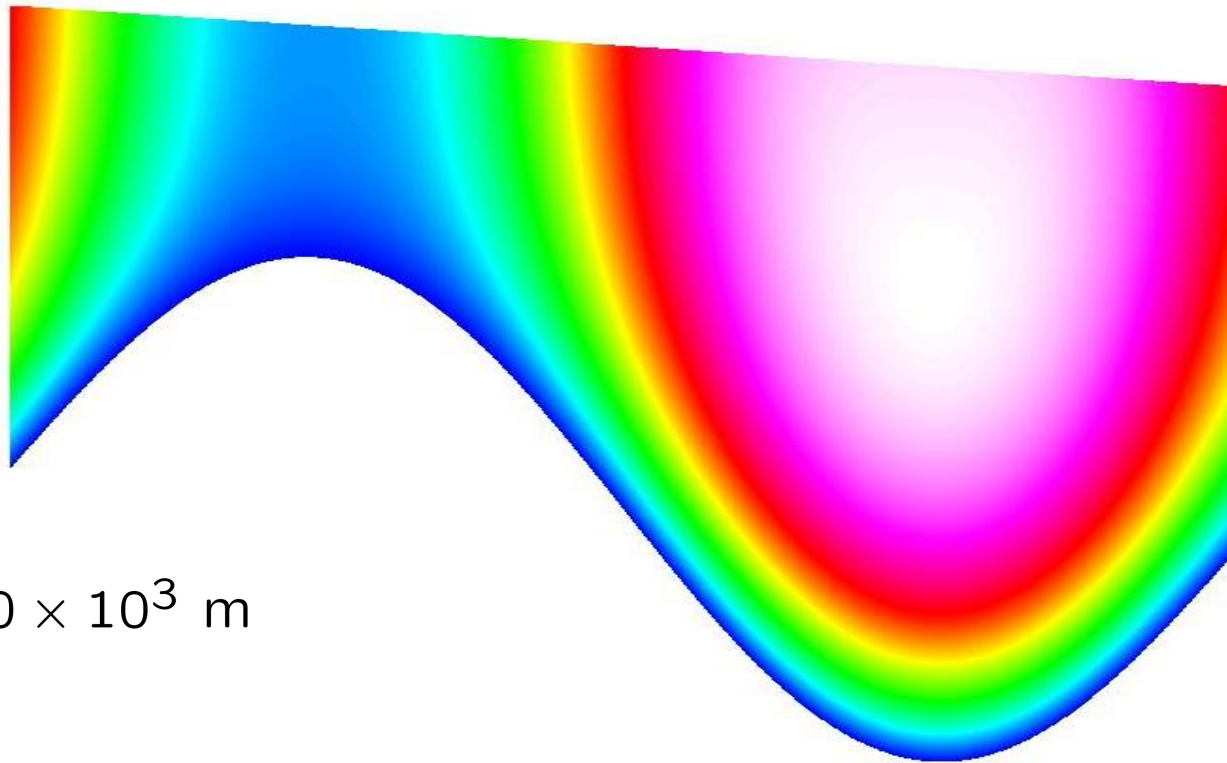


How to get a mesh ?

- ElmerGrid: the Elmer mesher
 - capability overview
 - importing mesh from other mesher
- Specific problems for glacier mesh
 - reserve of elements for growing glaciers
 - mesh coincide with the bed and surface data grid (case of synthetic data)
 - not structured mesh in the horizontal plane (bed and surface grid do not coincide with the mesh)
- Example: Different methods to mesh ISMIP-HOM B tests

Mesh of ISMIP-HOM B020 test

$$z_s(x) = -x \tan(0.5^\circ)$$



$$L = 20 \times 10^3 \text{ m}$$

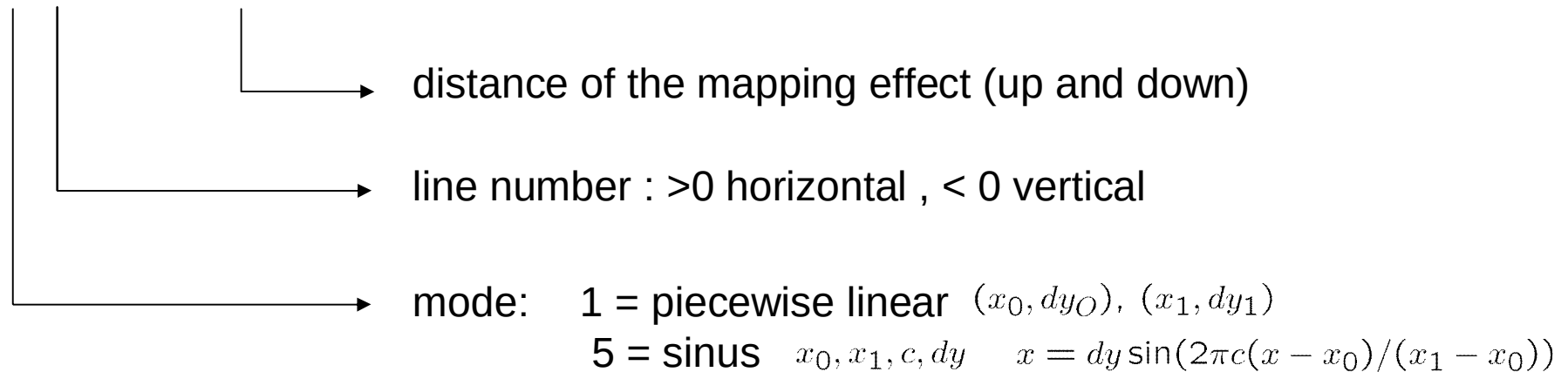
$$z_b(x) = z_s(x) - 1000 + 500 \sin\left(\frac{2\pi}{L}x\right)$$

Mesh with ElmerGrid

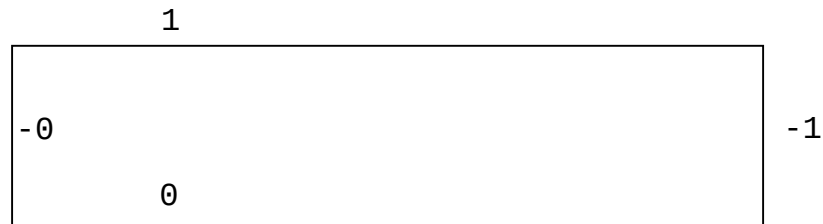
Use the boundary mappings of ElmerGrid (in mesh_B.grd)

```
...
Subcell Limits 1 = 0.0 2.000000000e+04
Subcell Limits 2 = -1000.0 0.0
```

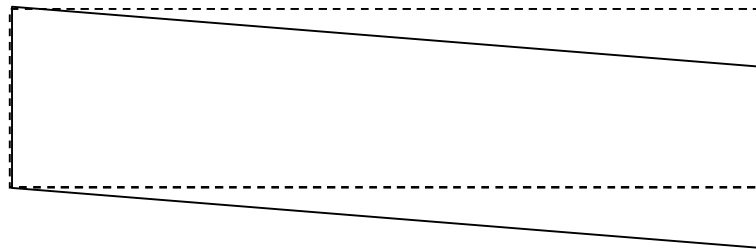
```
...
Geometry Mappings
! mode line limits(2) Np params(Np)
1 1 1000.0 1000.0 4 0.0 0.0 2.000000000e+04 -1.74537356e+02
1 0 1000.0 1000.0 4 0.0 0.0 2.000000000e+04 -1.74537356e+02
5 0 1000.0 1000.0 4 0.0 2.000000000e+04 1.0 500.0
End
```



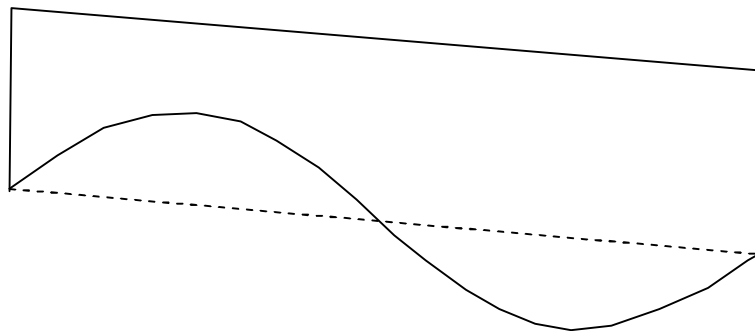
Mapping with ElmerGrid



```
1 1 1000.0 1000.0 4 0.0 0.0 2.000000000e+04 -1.74537356e+02
```



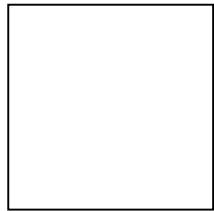
```
1 0 1000.0 1000.0 4 0.0 0.0 2.000000000e+04 -1.74537356e+02
```



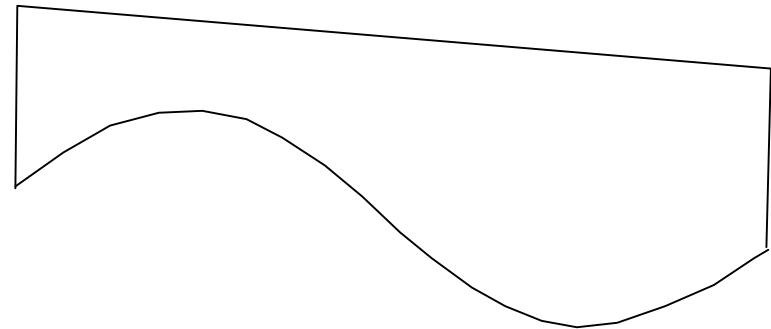
```
5 0 1000.0 1000.0 4 0.0 2.000000000e+04 1.0 500.0
```

Mesh with your own script

- Start from the square 1 x 1 of Step 0 and deform it to obtain the mesh of ISMIP B



The script of your choice !
(awk, f90, matlab, ...)



- Open the file mesh.nodes

- Read (x, y)

- Make the scaling :
$$\begin{cases} x \rightarrow L \times x \\ y \rightarrow z_b + (z_s - z_b) \times y \end{cases} \quad \begin{cases} z_s(x) = -x \tan(0.5^\circ) \\ z_b(x) = z_s(x) - 1000 + 500 \sin(\frac{2\pi}{L}x) \\ L = 20 \times 10^3 \text{ m} \end{cases}$$

- Re-write (x, y) in mesh.nodes

Mesh with your own script

Format of the file mesh.nodes :

Node number , -1, x, y, z (z=0 in 2D)

Examples:

- f90: see file MSH_ismipB.f90

```
PROGRAM MSH_ismipB
IMPLICIT NONE
REAL(KIND=8) :: x, y, z, xnew, ynew, zb, zs, L, Pi
REAL(KIND=8), ALLOCATABLE :: xnode(:), ynode(:)
CHARACTER :: NameMsh*20
INTEGER :: NtN, i, j, N
    Pi = ACOS(-1.0)
    WRITE(*,*)'Name of the Elmer mesh directory ?'
    READ(*,*)NameMsh
    WRITE(*,*)'Lenght of the mesh L [m] :'
    READ(*,*)L
    OPEN(10,file=TRIM(NameMsh)//"/mesh.header")
    READ(10,1000)NtN
    CLOSE(10)
    ALLOCATE(xnode(NtN), ynode(NtN))
    OPEN(12,file=TRIM(NameMsh)//"/mesh.nodes")
    READ(12,*)(N, j, xnode(i), ynode(i), z, i=1,NtN)
    REWIND(12)
```

```
DO N=1, NtN
    x = xnode(N)
    y = ynode(N)
    xnew = x * L
    zs = -xnew*TAN(0.5*Pi/180.0)
    zb = zs - 1000.0 + 500.0*SIN(2.0*Pi*xnew/L)
    ynew = zb + y * (zs - zb)
    WRITE(12,1200)N,j,xnew,ynew,z
END DO
DEALLOCATE(xnode, ynode)
1000 FORMAT(I6)
1200 FORMAT(i5,2x,i5,3(2x,e22.15))
END PROGRAM MSH_ismipB
```

- See other examples :

awk: see file script_awk

matlab: see file dilat_mesh.m

Mesh with your own mesher

Possible input format for ElmerGrid:

- 4) .ansys : Ansys input format
- 5) .inp : Abaqus input format by Ideas
- 6) .fil : Abaqus output format
- 7) .FDNEUT : Gambit (Fidap) neutral file
- 8) .unv : Universal mesh file format
- 9) .mphtxt : Comsol Multiphysics mesh format
- 10) .dat : Fieldview format
- 11) .node,.ele: Triangle 2D mesh format
- 12) .mesh : Medit mesh format
- 13) .msh : GID mesh format
- 14) .msh : Gmsh mesh format

Examples with gmsh: see file mesh_Bgmsh.geo

```
> gmsh mesh_Bgmsh.geo -2 -o mesh_Bgmsh.msh
```

```
> ElmerGrid 14 2 mesh_Bgmsh.msh
```

```
-> mesh_Bgmsh/mesh.*
```

```
> ElmerGrid 14 3 mesh_Bgmsh.msh
```

```
-> mesh_Bgmsh.ep (ElmerPost)
```