Basil

Finite Element Mesh viscous deformation code

Greg Houseman, University of Leeds
Terence Barr, AFEX Intl
Lynn Evans, University of Melbourne
(Philip England, Oxford University)

Basil uses FEM to calculate values which describe the stress and strain in a non-linear viscous fluid.

Deviatoric stress is defined using the gradients of the displacement rates:

$$\tau_{\alpha\beta} = 2\eta\dot{\varepsilon}_{\alpha\beta} = \eta\left(\frac{\partial u_\alpha}{\partial x_\beta} + \frac{\partial u_\beta}{\partial x_\alpha}\right)$$

Viscosity may vary spatially and with the local strain rate:

$$\eta = \frac{B}{2}\dot{E}^{\left(\frac{1}{n}-1\right)}$$

Where n is the non-Newtonian index and E is the second invariant of the strain tensor:

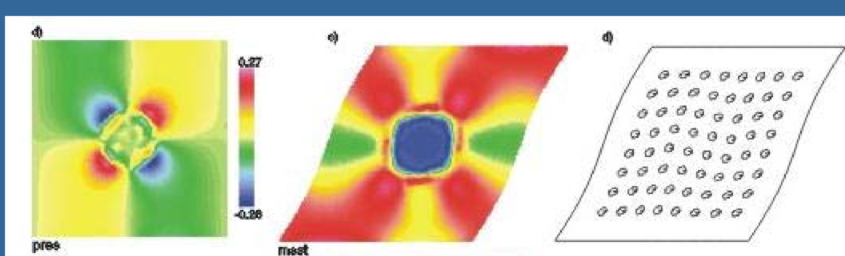
$$\dot{E} = \sqrt{\dot{\varepsilon}_{\alpha\beta}\dot{\varepsilon}_{\alpha\beta}}$$

Assemble stiffness matrix and load vectors, a and b, containing boundary stresses and body forces. Solve for u, v, p using a conjugate gradient solver

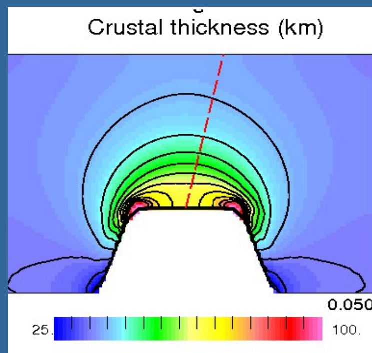
$$\begin{bmatrix} \mathbf{K}^{1} & \mathbf{K}^{4} & \mathbf{K}^{7} \\ \mathbf{K}^{4T} & \mathbf{K}^{2} & \mathbf{K}^{8} \\ \mathbf{K}^{7T} & \mathbf{K}^{8T} & \mathbf{0} \end{bmatrix} \begin{bmatrix} \mathbf{u} \\ \mathbf{v} \\ \mathbf{p} \end{bmatrix} = \begin{bmatrix} \mathbf{a} \\ \mathbf{b} \\ \mathbf{0} \end{bmatrix}$$

Conservation of momentum in a balance of the internal stress and the body force (gravity)
Conservation of mass

2D Plane Strain approximation assumes zero strain in the 3rd direction. The solution contains the 2 components of velocity (u,v) and the pressure field (p)



Thin Viscous Sheet approximation assumes that the stress in the 3rd direction is known and is usually gravitation PE with gravity acting perpendicular to the plane of calculation. Pressure is eliminated from the momentum equations and the solution contains u,v



Basil Input File

- SETUP – order is important as the command is processed when it is read
- TIMESTEP – set parameters controlling time intervals for calculations and saving solutions
- BOUNDARY conditions
- PHYSICAL parameters – control the distribution of viscosity contrast or density across the mesh

SETUP

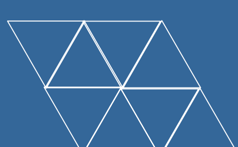
OUTPUT – set the following file options

- Directory for solution files, assumed to be FD.sols for *Elle* scripts
- Filename for solution record written after each timestep
- Filename for the binary file containing solution records
- Directory for text files, assumed to be FD.out for *Elle* scripts. BIN.out content is similar to screen output. BIN.dat content is controlled by SERIES command

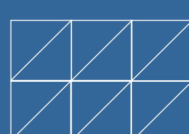
MESH

TYPE – 0,1,2 are regular *Basil* meshes

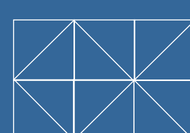
NX, NY – parameters controlling the number of elements



0

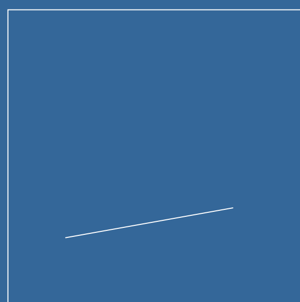


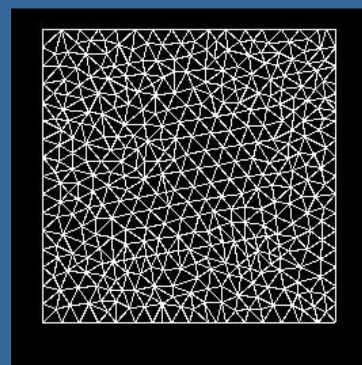
1



2

Type 3 is a triangulation of the nodes and segments in the POLY file





VISDENS – set physical parameters

•Viscosity

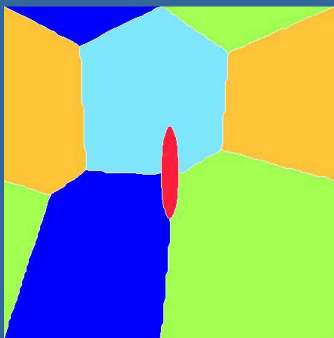
•Stress-strain exponent

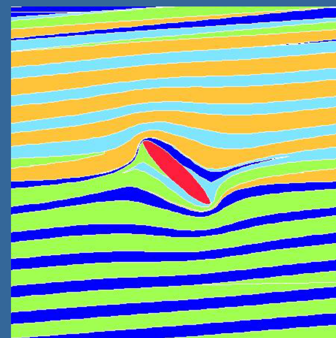
•Density

REG statements
A - apply to all elements
P - apply with the region defined by a coordinate list
E - apply to elements in a list

initial

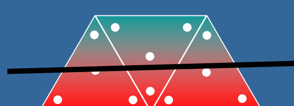
$\gamma = 12$





Hard ellipse in an homogeneous matrix
(grains colours allow differentiation)

Regular mesh



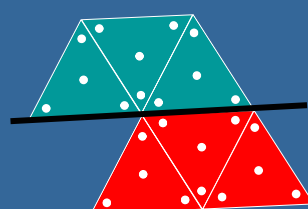
The viscosity coefficient is interpolated from the POLY file
REG A : VR = 10 ;

Find the polygons in the poly file

Each point is given the value of the polygon which contains it

Harder to find a solution
Inaccuracies if large contrast

Triangle mesh

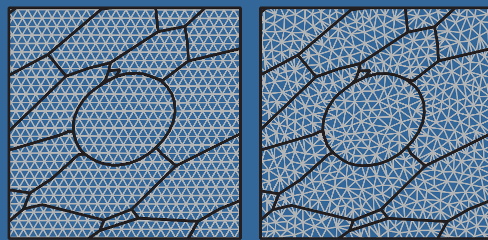


Triangle gives a value to each element within a polygon

Each point is given the value of the element which contains it

Contrast is between elements not within elements

Elle sets viscosity and SE using poly and Elle files



Regular mesh generated by *Basil*

Mesh generated by *Triangle*

MESH TYPE=0 NX=40 FAULT=1
POLY FILE=tmp.poly

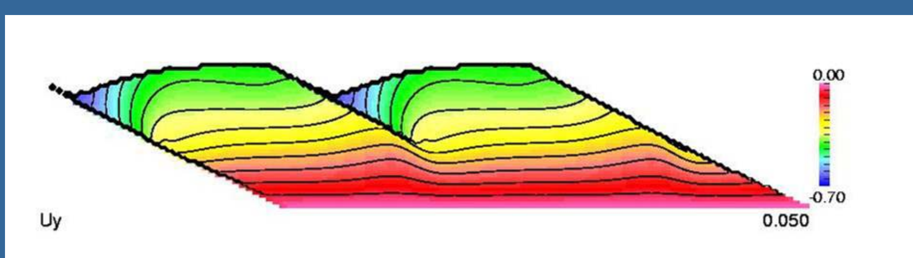
Values set at the 7 integration points by point-in-polygon test on the Elle flynns

MESH TYPE=3 FAULT=1
POLY FILE=tmp.poly

Values set at the 7 integration points by element

FAULTS (DISCONTINUITIES)

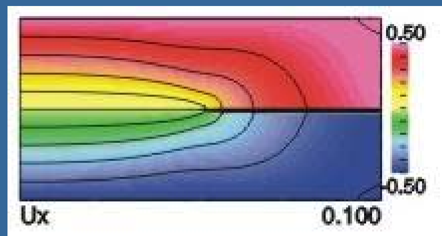
Introduce paired nodes on opposite sides of the fault
a set of stress variables in the solution.



Type 1 (periodic boundary)

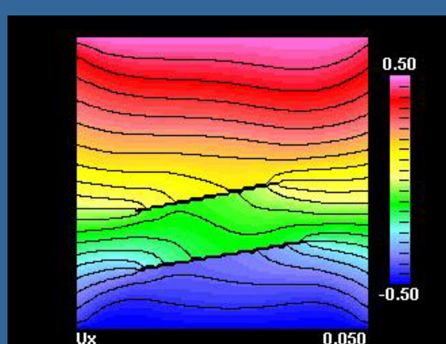
Fault always extends the length of the mesh.
Partially unlocked to model tiltblocks and shearbands
Elle uses a locked type 1 fault to model a torus

Type 2 - central internal fault (requires mesh 0-2)

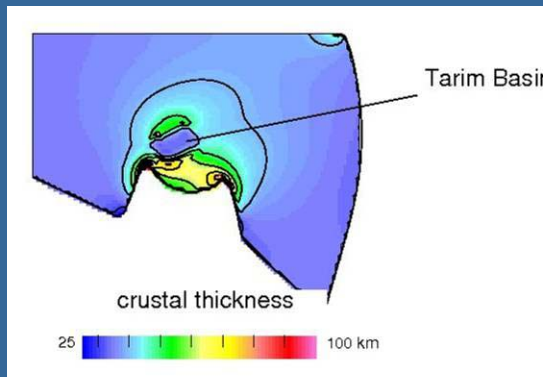


Partially unlocked to calculate pressure at a crack tip

Type 3 - general internal fault (requires type 3 mesh)



LAYER



TVS – shortening results in thickening.
Solution contains thickness arrays.
User specifies a horizontal length scale

SERIES

Data for specified nodes will be written to a text file (.dat extension) at every timestep

Data may be physical parameters or current position of a particular node or
min, max values of pressure, thickness for the mesh

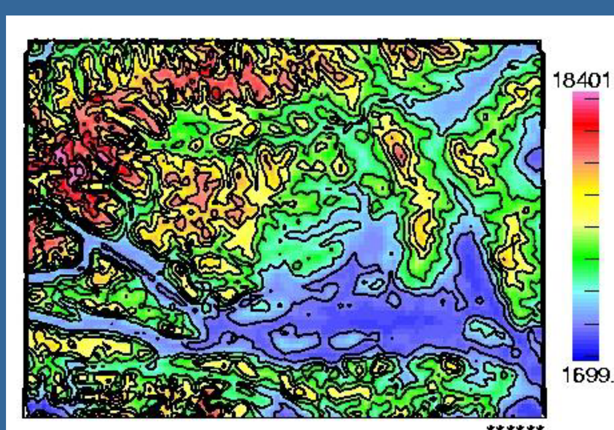
BCOND – set the boundary conditions

Horizontal and vertical velocities
General shear velocities
Traction
Friction coefficient
Fault – locking and unlocking

GIF Image

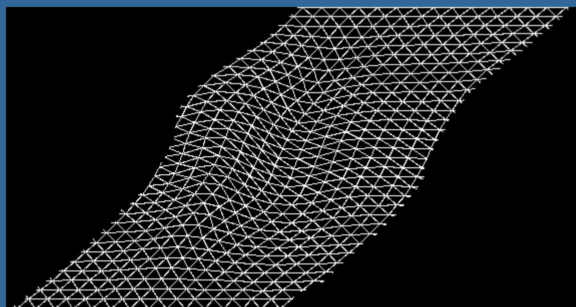
TOPO

Specify latitude and longitude range and a DEM file
Initialises thickness data



LAGRANGE MESH AND MARKERS

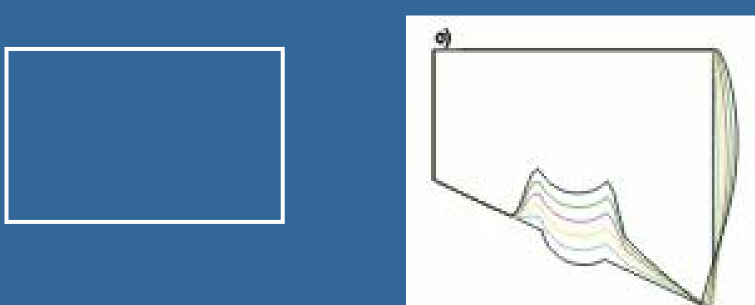
A simple mesh or grid of markers which passively advect with the medium and record the finite deformation



The markers are initially circular and can be analysed to determine finite strain ellipsoids

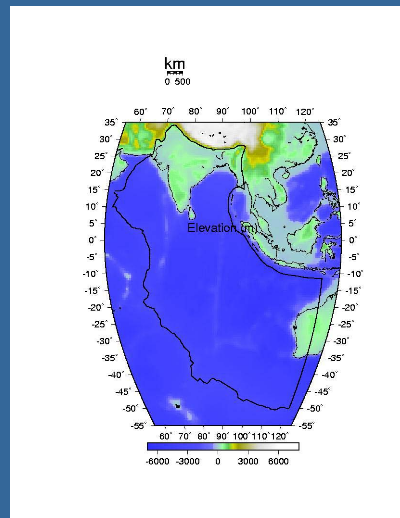
DEFORM

Deform one or more boundaries of a regular mesh



Deform a region within the mesh

Deform the whole mesh



TIMESTEP

• STEPSIZE – dt for calculations. May be reduced during a run if strain rate increases

Rt.gif

•STOP – no. of iterations and/or total time

• SAVE – time interval for saving solution records and a maximum no. of records

SOLVE

Parameters controlling the convergence of the solution at each timestep

Convergence criterion for linear solution and the non-linear iterations (n>1)
If these are set too small, round-off error may prevent convergence

Upper limit on the number of iterations

Elle2poly

- clip flynns against the unit cell – edge flynns may be divided into several polygons
- find an internal point for each polygon – the flynn number is the *Triangle* attribute
- add extra boundary nodes to ensure matching nodes along the external fault

Basil2elle

- Reads the *Elle* file (tmp.elle)
- Updates the bnode positions and unit cell
- Updates active strain attributes
- Writes a new *Elle* file with updated values and copies unchanged attributes