

Elmer Models Manual

Peter Råback, Mika Malinen, Juha Ruokolainen,
Antti Pursula, Thomas Zwinger, Eds.

CSC – IT Center for Science

May 29, 2019

Elmer Models Manual

About this document

Elmer Models Manual is a part of the documentation of Elmer finite element software. It consists of independent chapters describing different modules a.k.a. solvers which the main program (ElmerSolver) uses to create a specific physical model, although some solvers take care of computational tasks which are not tied up with the concept of physical model.

The structure of the manual reflects the modular architecture of the software which enables the addition of new modules without making any changes to the main program. Each chapter typically has a separate section for theory and keywords, and often some additional information is also given, for example about the limitations of the model. The Elmer Models Manual is best used as a reference manual rather than a concise introduction to the matter.

The present manual corresponds to Elmer software version 8.4. The latest documentation and program versions of Elmer are available (or links are provided) at <http://www.csc.fi/elmer>.

Copyright information

The original copyright of this document belongs to CSC – IT Center for Science, Finland, 1995–2019. This document is licensed under the Creative Commons Attribution-No Derivative Works 3.0 License. To view a copy of this license, visit <http://creativecommons.org/licenses/by-nd/3.0/>.

Elmer program is free software; you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation; either version 2 of the License, or (at your option) any later version. Elmer software is distributed in the hope that it will be useful, but without any warranty. See the GNU General Public License for more details.

Elmer includes a number of libraries licensed also under free licensing schemes compatible with the GPL license. For their details see the copyright notices in the source files.

All information and specifications given in this document have been carefully prepared by the best efforts of CSC, and are believed to be true and accurate as of time writing. CSC assumes no responsibility or liability on any errors or inaccuracies in Elmer software or documentation. CSC reserves the right to modify Elmer software and documentation without notice.

Contents

Table of Contents	3
I Models of Fluid Mechanics and Transport Phenomena	12
1 Heat Equation	13
1.1 Introduction	13
1.2 Theory	13
1.3 Keywords	15
2 Navier-Stokes Equations	22
2.1 Introduction	22
2.2 Theory	22
2.3 Keywords	26
Bibliography	31
3 Advection-Diffusion Equation	32
3.1 Introduction	32
3.2 Theory	32
3.3 Keywords	34
4 Advection-Reaction Equation	38
4.1 Introduction	38
4.2 Theory	38
4.3 Keywords	39
Bibliography	41
5 Reynolds Equation for Thin Film Flow	42
5.1 Introduction	42
5.2 Theory	42
5.3 Keywords	45
Bibliography	47
II Models of Solid Mechanics	48
6 Linear Elasticity	49
6.1 Introduction	49
6.2 Theory	49
6.3 Keywords	52

7	Finite Elasticity	56
7.1	Introduction	56
7.2	Field equations	56
7.3	Boundary conditions	58
7.4	Linearization: The basic constitutive laws	58
7.5	Linearization: The case of a user-defined material model	59
7.6	Stress and strain computation	60
7.7	Keywords	60
8	Shell Equations of Classical Elasticity	63
8.1	Introduction	63
8.2	Discrete shell model	64
8.3	Specifying surface data	69
8.4	Keywords	70
	Bibliography	71
9	Plate Equations of Linear Elasticity	72
9.1	Introduction	72
9.2	Finite element implementation	74
9.3	Elastic parameters for perforated plates	74
9.4	Keywords	75
	Bibliography	76
10	One-dimensional Equations for Elastic Beams	78
10.1	Introduction	78
10.2	Governing equations	78
10.3	The weak formulation	79
10.4	Keywords	80
III	Models of Acoustics	82
11	The Helmholtz Model	83
11.1	Introduction	83
11.2	Theory	83
11.3	Keywords	84
12	The Linearized Navier–Stokes Equations in the Frequency Domain	87
12.1	Introduction	87
12.2	Mathematical model	87
12.3	The use of block preconditioning	90
12.4	Utilities	91
12.5	Keywords	92
	Bibliography	95
13	Wave Equation	96
13.1	Introduction	96
13.2	Theory	96
13.3	Keywords	97
14	Large-amplitude Wave Motion in Air	99
14.1	Introduction	99
14.2	Mathematical model	99
14.3	Keywords	100

IV Models of Electromagnetism	102
15 Electrostatics	103
15.1 Introduction	103
15.2 Theory	103
15.3 Notes on output control	105
15.4 Keywords	105
16 Static Current Conduction	107
16.1 Introduction	107
16.2 Theory	107
16.3 Note on output control	108
16.4 Keywords	109
17 Computation of Magnetic Fields in 3D	110
17.1 Introduction	110
17.2 The transient equations	110
17.3 The stationary equations and the Helmholtz projection of a source	114
17.4 The boundary conditions	115
17.5 A consistently stabilized Lagrange multiplier formulation for magnetostatics	116
17.6 Keywords	116
18 Circuits and Dynamics Solver	122
18.1 Introduction	122
18.2 Theory	122
18.3 Keywords	124
Bibliography	126
19 Vectorial Helmholtz for Electromagnetic Waves	127
19.1 Introduction	127
19.2 Theory	127
19.3 Keywords	128
Bibliography	131
20 Computation of Magnetic Fields in 2D	132
20.1 Introduction	132
20.2 Theory	132
20.3 Keywords	133
21 Magnetic Induction Equation	136
21.1 Introduction	136
21.2 Theory	136
21.3 Keywords	137
22 Reduced Dimensional Electrostatics	140
22.1 Introduction	140
22.2 Theory	140
22.3 Implementation issues	142
22.4 Keywords	142
23 Poisson-Boltzmann Equation	144
23.1 Introduction	144
23.2 Theory	144
23.3 Notes on output control	146
23.4 Keywords	146
Bibliography	147

24	Loss Estimation Using the Fourier Series	148
24.1	Introduction	148
24.2	Loss estimation	149
24.3	Keywords	150
25	Coil Current Solver	153
25.1	Introduction	153
25.2	Theory	153
25.3	Keywords	155
V	Other Physical Models	158
26	Electrokinetics	159
26.1	Introduction	159
26.2	Theory	159
26.3	Limitations	160
26.4	Keywords	160
	Bibliography	161
27	Block Preconditioning for the Steady State Navier–Stokes Equations	163
27.1	Introduction	163
27.2	The model	163
27.3	Linearization	164
27.4	Discretization aspects	164
27.5	Block preconditioning	164
27.6	Defining additional solvers for subsidiary problems	165
27.7	Examples	166
27.8	Keywords	166
28	Rotational Form of the Incompressible Navier–Stokes Equations	169
28.1	Introduction	169
28.2	Field equations	169
28.3	Boundary conditions	170
28.4	Linearization	170
28.5	Discretization aspects	171
28.6	Utilizing splitting strategies by preconditioning	171
28.7	Restrictions	172
28.8	Keywords	172
VI	Free Surfaces, Phase Change and Particle Dynamics	175
29	Level-Set Method	176
29.1	Introduction	176
29.2	Theory	176
29.3	Keywords	178
30	Kinematic Free Surface Equation with Limiters	182
30.1	Introduction	182
30.2	Theory	182
30.3	Constraints	183
30.4	Keywords	184

31 Free Surface with Constant Flux	186
31.1 Introduction	186
31.2 Theory	186
31.3 Applicable cases and limitations	187
31.4 Keywords	187
32 Transient Phase Change Solver	189
32.1 Introduction	189
32.2 Theory	189
32.3 Applicable cases and limitations	190
32.4 Keywords	190
33 Steady State Phase Change Solver	193
33.1 Introduction	193
33.2 Theory	193
33.3 Applicable cases and limitations	194
33.4 Keywords	195
34 Particle Dynamics	197
34.1 Introduction	197
34.2 Theory	198
34.3 Keywords	200
35 Semi-Lagrangian Advection Using Particle Tracking	205
35.1 Introduction	205
35.2 Theory	205
35.3 Parallel operation	206
35.4 Keywords	206
VII Mesh Adaptation and Transformation	208
36 Mesh Adaptation Solver	209
36.1 Introduction	209
36.2 Theory	209
36.3 Keywords	210
36.4 Examples	211
37 Nonphysical Mesh Adaptation Solver	212
37.1 Introduction	212
37.2 Theory	212
37.3 Keywords	212
38 Rigid Mesh Transformation	214
38.1 Introduction	214
38.2 Theory	214
38.3 Keywords	215
39 Structured Mesh Mapper	217
39.1 Introduction	217
39.2 Theory	217
39.3 Keywords	217

VIII Utility Solvers	220
40 Streamline Computation	221
40.1 Introduction	221
40.2 Theory	221
40.3 Limitations	222
40.4 Keywords	222
40.5 Example	223
41 Flux Computation	224
41.1 Introduction	224
41.2 Theory	224
41.3 Implementation issues	224
41.4 Keywords	225
41.5 Example	226
42 Vorticity Computation	227
42.1 Introduction	227
42.2 Theory	227
42.3 Keywords	227
42.4 Example	228
43 Divergence Computation	229
43.1 Introduction	229
43.2 Theory	229
43.3 Keywords	229
44 Scalar Potential Resulting to a Given Flux	231
44.1 Introduction	231
44.2 Theory	231
44.3 Keywords	231
45 System Reduction for Displacement Solvers	233
45.1 Introduction	233
45.2 Theory	233
45.3 Applicable cases and limitations	234
45.4 Keywords	234
45.5 Examples	235
46 Artificial Compressibility for FSI	237
46.1 Introduction	237
46.2 Theory	237
46.3 Keywords	240
Bibliography	241
47 Internal Cost Function Optimization	243
47.1 Introduction	243
47.2 Theory	243
47.3 Keywords	243
IX Saving Data and Results	246
48 Saving Scalar Values to a File	247
48.1 Introduction	247
48.2 Theory	247

48.3	Implementation issues	248
48.4	Keywords	248
48.5	Examples	251
49	Saving data along lines to a file	253
49.1	Introduction	253
49.2	Theory	253
49.3	Keywords	253
49.4	Examples	255
50	Saving material parameters and boundary conditions	256
50.1	Introduction	256
50.2	Keywords	256
51	Result output in different formats	258
51.1	Introduction	258
51.2	Keywords	258
51.3	Example	261
52	Saving data on uniform cartesian grid	262
52.1	Introduction	262
52.2	Keywords	262
52.3	Example	264
53	Isosurface extraction for reduced output	265
53.1	Introduction	265
53.2	Keywords	265
53.3	Example	266
54	Coupling Elmer with OpenFOAM via file IO	267
54.1	Introduction	267
54.2	Keywords for module Elmer2OpenFoamIO	267
54.3	Keywords for module OpenFoam2ElmerIO	268
54.4	Example	269
55	Reload Existing Simulation Results	270
55.1	Introduction	270
55.2	Keywords	270
56	Runtime Control of the Input Data	272
56.1	Introduction	272
56.2	Limitations	272
56.3	Keywords	272
57	Reading NetCDF data into FE mesh	273
57.1	Introduction	273
57.2	Theory	273
57.3	Keywords	274
58	Filtering Time-Series Data	276
58.1	Introduction	276
58.2	Theory	276
58.3	Keywords	277

59 Data to field solver	279
59.1 Introduction	279
59.2 Theory	279
59.3 Keywords	280
60 Projection to plane	282
60.1 Introduction	282
60.2 Theory	282
60.3 Keywords	283
61 Structured projection to plane	284
61.1 Introduction	284
61.2 Theory	284
61.3 Keywords	284
62 Statistics of finite element mesh	286
62.1 Introduction	286
62.2 Keywords	286
X Experimental or Obsolete Solvers	287
63 Richards equation for variably saturated porous flow	288
63.1 Introduction	288
63.2 Theory	288
63.3 Implementation issues	289
63.4 Keywords	289
64 BEM Solver for Poisson Equation	292
64.1 Introduction	292
64.2 Theory	292
64.3 Keywords	293
65 BEM Solver for Helmholtz Equation	295
65.1 Introduction	295
65.2 Theory	295
65.3 Keywords	296
66 Outlet Boundary Condition for Arterial Flow Simulations	298
66.1 Introduction	298
66.2 Theory	298
66.3 Keywords	299
Bibliography	301
67 Magnetoquasistatic approximation for axial symmetry	302
67.1 Introduction	302
67.2 Theory	302
67.3 Keywords	303
68 Electrostatics of Moving Rigid Bodies	306
68.1 Introduction	306
68.2 Electrostatics	306
68.3 Mesh movement	307
68.4 Implimentation issues	308
68.5 Keywords	308

69 Fluidic Force	310
69.1 Introduction	310
69.2 Theory	310
69.3 Additional output	311
69.4 Keywords	311
70 Electrostatic force	312
70.1 Introduction	312
70.2 Theory	312
70.3 Keywords	312
Bibliography	313
71 Linear Constraints	314
71.1 Introduction	314
71.2 Theory	314
71.3 Limitations	314
71.4 Keywords	315
Bibliography	315
72 Density Functional Theory	316
72.1 Introduction	316
72.2 Theory	316
72.3 Keywords	317
Bibliography	319
73 Parallel I/O using HDF5 library	320
73.1 Introduction	320
73.2 Keywords	320
73.3 Example	321
Index	322

Part I

**Models of Fluid Mechanics and
Transport Phenomena**

Model 1

Heat Equation

Module name: HeatSolve

Module subroutines: HeatSolver

Module authors: Juha Ruokolainen, Peter Råback, Matthias Zenker

Document authors: Juha Ruokolainen, Ville Savolainen, Peter Råback

Document edited: July 29th 2002

1.1 Introduction

Heat equation results from the requirement of energy conservation. In addition the Fourier's law is used to model the heat conduction. The linearity of the equation may be ruined by temperature dependent thermal conductivity, or by heat radiation.

1.2 Theory

1.2.1 Governing Equations

The incompressible heat equation is expressed as

$$\rho c_p \left(\frac{\partial T}{\partial t} + (\vec{u} \cdot \nabla) T \right) - \nabla \cdot (k \nabla T) = \bar{\bar{\tau}} : \bar{\bar{\varepsilon}} + \rho h, \quad (1.1)$$

where ρ is the density, c_p the heat capacity at constant pressure, T the temperature, \vec{u} the convection velocity, k the heat conductivity and h is source of heat. The term $\bar{\bar{\tau}} : \bar{\bar{\varepsilon}}$ is the frictional viscous heating, which is negligible in most cases. For Newtonian fluids, the viscous part of the stress tensor is

$$\bar{\bar{\tau}} = 2\mu\bar{\bar{\varepsilon}}, \quad (1.2)$$

where $\bar{\bar{\varepsilon}}$ the linearized strain rate tensor.

Eq.1.1 applies also for solids, setting $\vec{u} = 0$. For solids, conduction may be anisotropic and the conductivity a tensor.

For compressible fluids, the heat equation is written as

$$\rho c_v \left(\frac{\partial T}{\partial t} + \vec{u} \cdot \nabla T \right) - \nabla \cdot (k \nabla T) = -p \nabla \cdot \vec{u} + \bar{\bar{\tau}} : \bar{\bar{\varepsilon}} + \rho h, \quad (1.3)$$

where c_v is the heat capacity at constant volume. The density needs to be calculated from the equation of state, e.g., perfect gas law. More information is given in the chapter describing the Navier-Stokes equation.

The Elmer heat equation module is capable of simulation heat transfer by conduction, convection, and diffuse gray radiation. Also a phase change model is included. Couplings to other modules include, convection by fluid flow, frictional heating (modules providing flow fields), and resistive heating (modules providing magnetic and/or electric fields).

1.2.2 Arbitrary Lagrangian-Eulerian (ALE) coordinates

For problems involving a deforming mesh the transient heat equation must be solved using Arbitrary Lagrangian-Eulerian (ALE) frame of reference. Assume that the mesh velocity is \vec{c} . Then the convective term yields

$$\rho c_p ((\vec{u} - \vec{c}) \cdot \nabla) T \quad (1.4)$$

1.2.3 Phase Change Model

Elmer has an internal fixed grid phase change model. Modelling phase change is done by modifying the definition of heat capacity according to whether a point in space is in solid or liquid phase or in a 'mushy' region. The choice of heat capacity within the intervals is explained in detail below.

This type of algorithm is only applicable, when the phase change occurs within finite temperature interval. If the modelled material is such that the phase change occurs within very sharp temperature interval, this method might not be appropriate.

For the solidification phase change model Elmer uses, we need enthalpy. The enthalpy is defined to be

$$H(T) = \int_0^T \left(\rho c_p + \rho L \frac{\partial f}{\partial \lambda} \right) d\lambda, \quad (1.5)$$

where $f(T)$ is the fraction of liquid material as a function of temperature, and L is the latent heat. The enthalpy-temperature curve is used to compute an effective heat capacity, whereupon the equations become identical to the heat equation. There are two ways of computing the effective heat capacity in Elmer:

$$c_{p,\text{eff}} = \frac{\partial H}{\partial T}, \quad (1.6)$$

and

$$c_{p,\text{eff}} = \left(\frac{\nabla H \cdot \nabla H}{\nabla T \cdot \nabla T} \right)^{1/2}. \quad (1.7)$$

The former method is used only if the local temperature gradient is very small, while the latter is the preferred method. In transient simulations a third method is used, given by

$$c_{p,\text{eff}} = \frac{\partial H / \partial t}{\partial T / \partial t}. \quad (1.8)$$

Note that for the current implementation of the heat equation heat capacity and enthalpy are additive. So if heat capacity is present in the command file it should not be incorporated to enthalpy as an integral.

1.2.4 Additional Heat Sources

Frictional heating is calculated currently, for both incompressible and compressible fluids, by the heat source

$$h_f = 2\mu \bar{\bar{\epsilon}} : \bar{\bar{\epsilon}}. \quad (1.9)$$

In case there are currents in the media the also the the resistive heating may need to be considered. The Joule heating is then given by

$$h_m = \frac{1}{\sigma} \vec{J} \cdot \vec{J}. \quad (1.10)$$

In the above equations, \vec{B} and \vec{E} are the magnetic and electric fields, respectively. The current density \vec{J} is defined as

$$\vec{J} = \sigma(\vec{E} + \vec{u} \times \vec{B}). \quad (1.11)$$

In modeling biological tissue perfused with blood acting as heat sink an additional heat source term of the Pennes' Bioheat equation is needed. The term is

$$h_b = c_b \rho_b w (T_b - T) \quad (1.12)$$

where c_b is the specific heat capacity, ρ_b the density, and T_b the temperature of the blood. The perfusion rate w is the volume of blood flowing through a unit volume of tissue per second. This additional source term is modeled so that the part including T is treated implicitly for better convergence. Even though the model was written for the biological application in mind the additional heat source may find also other uses.

1.2.5 Boundary Conditions

For temperature one can apply boundary conditions and have either temperature or heat flux prescribed.

Dirichlet boundary condition (temperature is prescribed) reads as

$$T = T_b. \quad (1.13)$$

The value of T_b can be constant or a function of time, position or other variables.

Heat flux depending on heat transfer coefficient α and external temperature T_{ext} may be written as

$$-k \frac{\partial T}{\partial n} = \alpha(T - T_{\text{ext}}). \quad (1.14)$$

Both variables α and T_{ext} can be constant or functions of time, position or other variables. If the heat transfer coefficient α is equal to zero, it means that the heat flux on a boundary is identically zero. The Neumann boundary condition $-k\partial T/\partial n = 0$ is also used in a symmetry axis in 2D, axisymmetric or cylindrical problems.

Heat flux can consist of idealized radiation whereupon

$$-k \frac{\partial T}{\partial n} = \sigma \varepsilon (T^4 - T_{\text{ext}}^4). \quad (1.15)$$

Above, σ is the Stefan-Boltzmann constant and ε the surface emissivity. The emissivity and the external temperature can again be constant or functions of time, position, or other variables.

If the surface k is receiving radiation from other surfaces in the system, then the heat flux reads as

$$-k_k \frac{\partial T_k}{\partial n_k} = \sigma \varepsilon_k (T_k^4 - \frac{1}{A_k \varepsilon_k} \sum_{i=1}^N G_{ik} \varepsilon_i T_i^4 A_i), \quad (1.16)$$

where the subscripts i and k refer to surfaces i and k , and the parameters A_i and A_k to the specific surface areas. The factors G_{ik} are Gebhardt factors, and N represents the total number of radiating surfaces present in the system. Emissivities are assumed to be constant on each surface.

The heat equation is nonlinear when radiation is modelled. The nonlinear term in the boundary condition (1.15) can be linearized as

$$T^4 - T_{\text{ext}}^4 \approx (\mathcal{T}^3 + T_{\text{ext}} \mathcal{T}^2 + T_{\text{ext}}^2 \mathcal{T} + T_{\text{ext}}^3)(T - T_{\text{ext}}), \quad (1.17)$$

where \mathcal{T} is the temperature from the previous iteration.

One may also give an additional heat flux term as

$$-k \frac{\partial T}{\partial n} = q. \quad (1.18)$$

1.3 Keywords

Constants

Stefan Boltzmann `Real`

The value of the Stefan-Boltzmann constant needed for thermal radiation.

Simulation

The simulation section gives the case control data:

Simulation Type `String`

Heat equation may be either Transient or Steady State.

Coordinate System `String`

Defines the coordinate system to be used, one of: Cartesian 1D, Cartesian 2D, Cartesian 3D, Polar 2D, Polar 3D, Cylindric, Cylindric Symmetric and Axi Symmetric.

`Timestepping Method String`

Possible values of this parameter are Newmark (an additional parameter `Newmark Beta` must be given), BDF (BDF `Order` must be given). Also as a shortcut to Newmark-method with values of `Beta`= 0.0, 0.5, 1.0 the keywords `Explicit Euler`, `Crank-Nicolson`, and `Implicit Euler` may be given respectively. The recommended choice for the first order time integration is the BDF method of order 2.

`BDF Order Integer`

Value may range from 1 to 5.

`Newmark Beta Real`

Value in range from 0.0 to 1.0. The value 0.0 equals to the explicit Euler integration method and the value 1.0 equals to the implicit Euler method.

`Solver solver id`

The solver section defines equation solver control variables. Most of the possible keywords – related to linear algebra, for example – are common for all the solvers and are explained elsewhere.

`Equation String Heat Equation`

The name of the equation.

`Nonlinear System Convergence Tolerance Real`

The criterion to terminate the nonlinear iteration after the relative change of the norm of the field variable between two consecutive iterations is small enough

$$||T_i - T_{i-1}|| < \epsilon ||T_i||,$$

where ϵ is the value given with this keyword.

`Nonlinear System Max Iterations Integer`

The maximum number of nonlinear iterations the solver is allowed to do.

`Nonlinear System Newton After Iterations Integer`

Change the nonlinear solver type to Newton iteration after a number of Picard iterations have been performed. If a given convergence tolerance between two iterations is met before the iteration count is met, it will switch the iteration type instead. In the heat equation the Picard iterations means that the radiation term is factorized to linear and third-power terms.

`Nonlinear System Newton After Tolerance Real`

Change the nonlinear solver type to Newton iteration, if the relative change of the norm of the field variable meets a tolerance criterion:

$$||T_i - T_{i-1}|| < \epsilon ||T_i||,$$

where ϵ is the value given with this keyword.

`Nonlinear System Relaxation Factor Real`

Giving this keyword triggers the use of relaxation in the nonlinear equation solver. Using a factor below unity is sometimes required to achieve convergence of the nonlinear system. A factor above unity might speed up the convergence. Relaxed variable is defined as follows:

$$T'_i = \lambda T_i + (1 - \lambda) T_{i-1},$$

where λ is the factor given with this keyword. The default value for the relaxation factor is unity.

`Steady State Convergence Tolerance Real`

With this keyword a equation specific steady state or coupled system convergence tolerance is given. All the active equation solvers must meet their own tolerances before the whole system is deemed converged. The tolerance criterion is:

$$||T_i - T_{i-1}|| < \epsilon ||T_i||,$$

where ϵ is the value given with this keyword.

Stabilize Logical

If this flag is set true the solver will use stabilized finite element method when solving the heat equation with a convection term. If this flag is set to `False` RFB (Residual Free Bubble) stabilization is used instead (unless the next flag `Bubbles` is set to `False` in a problem with Cartesian coordinate system). If convection dominates stabilization must be used in order to successfully solve the equation. The default value is `False`.

Bubbles Logical

There is also a residual-free-bubbles formulation of the stabilized finite-element method. It is more accurate and does not include any ad hoc terms. However, it may be computationally more expensive. The default value is `True`. If both `Stabilize` and `Bubbles` are set to `False`, no stabilization is used. Note that in this case, the results might easily be nonsensical.

Smart Heater Control After Tolerance Real

The smart heater control should not be activated before the solution has somewhat settled. By default the smart heater control is set on when the Newtonian linearization is switched on for the temperature equation. Sometimes it may be useful to have more stringent condition for turning on the smart heater control and then this keyword may be used to give the tolerance.

Apply Limiter Logical

The generic soft limiters may be applied for the heat equation equation. They could for example, account for the effects of phase change under circumstances where it may be assumed that the temperature does not go over the phase change temperature. With this flag active the minimum and maximum limiters are accounted.

In some cases the geometry or the emissivities of the radiation boundaries change. This may require the recomputation of the view factors and Gebhardt factors. For that purpose also dynamic computation of the factors is enabled and it is controlled by the keywords below. The radiation factors are also automatically computed if no files for the factors are given although radiation boundaries exist.

Update View Factors Logical

The recomputation of the view factors is activated by setting the value of this flag to `True`. `False` is the default.

Update Gebhardt Factors Logical

If the emissivities depend on the solution the Gebhardt factors may need to be recomputed. This is activated by setting giving this flag value `True`. `False` is the default.

Minimum View Factor Real

This keyword determines the cut-off value under which the view factors are omitted. Neglecting small values will not only save memory but also will make the matrix used for solving the Gebhardt factors less dense. This consequently will enable more efficient sparse matrix strategies in solving the Gebhardt factors. The value for this parameter might be of the order 10^{-8} .

Minimum Gebhardt Factor Real

The Gebhardt factors make part of matrix dense. By neglecting the smallest Gebhardt factors the matrix structure for the heat equation may become significantly sparser and thus the solution time may drop. The value for this parameter might also be of the order 10^{-8} .

Implicit Gebhardt Factor Fraction Real

In computing heat transfer problems with radiation in an implicit manner the matrix structure becomes partially filled. This affects the performance of the linear equation solvers and also increases the memory requirements. On the other hand explicit treatment of radiation slows down the convergence significantly. This keyword allows that the largest Gebhardt factors are treated in an implicit manner whereas the smallest are treated explicitly. The value should lie in between zero (fully explicit) and one (fully implicit).

Matrix Topology Fixed Logical

If the Gebhardt factors change the matrix structure of the heat equation may also have to be changed unless this flag is set to `False`. Then all factors that do not combine with the matrix structure are omitted.

View Factors Geometry Tolerance Real

The view factors take a lot of time to compute. Therefore during the iteration a test is performed to check whether the geometry has changed. If the relative maximum change in the coordinate values is less than the value given by this parameter the view factors are not recomputed and the old values are used.

View Factors Fixed After Iterations Integer

Sometimes the iteration changes the geometry of the radiation boundaries as an unwanted side-effect. Then the geometry on the radiation boundary may be set fixed after some iterations. In practice this is done by adding suitable Dirichlet conditions in the boundary conditions.

Gebhardt Factors Fixed After Iterations Integer

Sometimes the emissivity depends on temperature but recomputing it every time may be costly. By this keyword the recomputation may be limited to the given number of visits to the heat equation solver.

View Factors Fixed Tolerance Real

This keywords defines the coupled system tolerance for the heat equation after which the recomputation of view factors is omitted. Typically this should be defined by a geometry tolerance but if the temperature solver follows the changes in geometry this may be a good control as well.

Gebhardt Factors Fixed Tolerance Real

This keywords defines the coupled system tolerance for the heat equation after which the recomputation of Gebhardt factors is omitted. The temperature dependence of emissivity is typically not so strong that small temperature changes would result to a need to recompute the Gebhardt factors as well.

Gebhardt Factors Solver Full Logical

If the view factor matrix is relatively sparse it will make sense to use a sparse matrix equation for solving the Gebhardt factors. This flag may be used if a full matrix should be desired.

Gebhardt Factors Solver Iterative Logical

If the Gebhardt factors are solved from a sparse matrix equation also the type of solver may be selected. The default is direct umfpack solver. Sometimes the memory usage may be a problem or the direct strategy simply not efficient enough. Then an iterative cgs solver may be used instead.

radiation: Linear System Keyword

Sometimes the default routines for computing Gebhardt factors are not suitable. For that purposes the user may specify any linear system strategy using namespace `radiation:` and the standard linear system keywords. The solver instance is created on the fly and the keywords are passed to it without the suffix.

Viewfactor Divide Integer

For axisymmetric view factor computation gives the number of divisions for each element. The default is 1.

Viewfactor Combine Elements Logical

There may be a significant amount of saved time if in the axisymmetric view factor computation the elements that are aligned and share a common node are united. The shadowing loop will then only be performed over these macroelements.

Equation eq id

The equation section is used to define a set of equations for a body or set of bodies.

Heat Equation String

If set to True, solve the heat equation.

Convection String

The type of convection to be used in the heat equation, one of: None, Computed, Constant.

Phase Change Model String

One of: None, Spatial 1, Spatial 2 and Temporal. Note that when solidification

is modelled, the enthalpy-temperature- and viscosity-temperature-curves must be defined in the material section.

Body Forces `bf id`

The body force section may be used to give additional force terms for the equations. The following keywords are recognized by the base solver:

Heat Source `Real`

A heat source h for the heat equation may be given with this keyword. Note that by default the heating is given per unit mass, not unit volume.

Friction Heat `Logical`

Currently redundant keyword, the frictional heating h_f is automatically added.

Joule Heat `Logical`

If set `True`, triggers use of the electromagnetic heating. This keywords accounts for the heating of many different solvers; electrostatics, magnetostatics, and induction equation.

Smart Heater Control `Logical`

Sometimes the prescribed heat source does not lead to the desired temperature. Often the temperature is controlled by a feedback and therefore a similar heater control in the simulation may give more realistic results. This flag makes sets the smart heater control on for the given body force.

Integral Heat Source `Real`

This keyword activates a normalization of the `Heat Source` so that the integral heating power is the desired objective.

Temperature Lower Limit `Real`

The lower limit for temperature that is enforced iteratively when the soft limiters are applied.

Temperature Upper Limit `Real`

The upper limit for temperature that is enforced iteratively when the soft limiters are applied.

There are four optional keywords related to the Pennes' bioheat equation term that model the perfusion process.

Perfusion Rate `Real`

The rate of the perfusion w . Activates the perfusion process.

Perfusion Reference Temperature `Real`

Temperature T_b of the perfusion fluid.

Perfusion Density `Real`

Density ρ_b of the perfusion fluid.

Perfusion Heat Capacity `Real`

Heat capacity c_b of the perfusion fluid.

Initial Condition `ic id`

The initial condition section may be used to set initial values for temperature.

Temperature `Real`

Material `mat id`

The material section is used to give the material parameter values. The following material parameters may be effective when heat equation is solved.

Density `Real`

The value of density is given with this keyword. The value may be constant, or variable. For the compressible flow, the density is computed internally, and this keyword has no effect.

Enthalpy `Real`

Note that, when using the solidification modelling, an enthalpy-temperature curve must be given. The enthalpy is derived with respect to temperature to get the value of the effective heat capacity.

Viscosity Real

Viscosity is needed if viscous heating is taken into account. When using the solidification modelling, a viscosity-temperature curve must be given. The viscosity must be set to high enough value in the temperature range for solid material to effectively set the velocity to zero.

Heat Capacity Real

The value of heat capacity in constant pressure c_p is given with this keyword. The value may be constant, or variable. For the phase change model, this value is modified according to rules given in the theory section.

Heat Conductivity Real

The value of heat conductivity k is given with this keyword. The value may be a constant or variable.

Convection Velocity i Real

Convection velocity $i=1, 2, 3$ for the constant convection model.

Compressibility Model Real

This setting may be used to set the compressibility model for the flow simulations. Choices are Incompressible and Perfect Gas. If set to the latter there may be mechanical work performed by the heating. Then also the settings Reference Pressure and Specific Heat Ratio must also be given.

Reference Pressure Real

With this keyword a reference level of pressure may be given.

Specific Heat Ratio Real

The ratio of specific heats (in constant pressure versus in constant volume) may be given with this keyword. The default value of this setting is 5/3, which is the appropriate value for monoatomic ideal gas.

Emissivity Real

Emissivity of the radiating surface, required for radiation model is present. If the emissivity is not found in the radiating boundary only then will it be looked at the material properties of the parent elements. Often locating the emissivity here makes the case definition more simple.

Transmissivity Real

For the diffuse gray radiation model also transmissivity of the surface may be provided. It gives the part of the energy that is lost as it passes through the wall. By default transmissivity is zero.

Boundary Condition bc id

The boundary condition section holds the parameter values for various boundary condition types. In heat equation we may set the temperature directly by Dirichlet boundary conditions or use different flux conditions for the temperature. The natural boundary condition of heat equation is zero flux condition.

Temperature Real**Heat Flux BC** Logical

Must be set to True, if heat flux boundary condition is present.

Heat Flux Real

A user defined heat flux term.

Heat Transfer Coefficient Real

Defines the parameter α in the heat flux boundary condition of the type

$$-k \frac{\partial T}{\partial n} = \alpha(T - T_{ext}).$$

External Temperature Real

Defines the variable for ambient temperature T_{ext} in the previous equation.

Radiation String

The type of radiation model for this boundary, one of: None, Idealized, Diffuse Gray. Note that, when using the diffuse gray radiation model, the file containing the Gebhardt factors must be given in the simulation section.

Radiation Boundary Integer

If there are many closures with radiation boundary conditions that do not see each other the view factors may be computed separately. This keyword is used to group the boundaries to independent sets. The default is one.

Radiation Boundary Open Logical

The closures may be partially open. Then no normalization of the view factors is enforced. The missing part of the radiation angle is assumed to be ideal radiation. Therefore if this option is enforced also the parameter External Temperature must be given.

Radiation External Temperature Real

In case the external temperature related to the heat transfer coefficient is different than that related to the radiation they cannot be given with the same keyword. For this purpose an alternative keyword is provided for radiation problems. This is used instead if it present.

Emissivity Real

Emissivity of the radiating surface, required for radiation model is present. If the emissivity is not found here it will be searched at the parent elements.

Transmissivity Real

If the transmissivity is not found here it will be searched at the parent elements.

Radiation Target Body Integer

This flag may be used to set the direction of the outward pointing normal. This is used when computing viewfactors. A body identification number must be given. The default is that the normal points to less dense material or outward on outer boundaries.

Smart Heater BoundaryLogical If the smart heater is activated the point for monitoring the temperature is the point with maximum x -coordinate on the boundary where this keyword is set True. Alternatively the logical variable **Phase Change** is looked for.

Smart Heater TemperatureReal The desired temperature for the smart heater system is set by this keyword. Alternatively the real variable **Melting Point** may be used.

Model 2

Navier-Stokes Equations

Module name: FlowSolve

Module subroutines: FlowSolver

Module authors: Juha Ruokolainen

Document authors: Juha Ruokolainen, Peter Råback

Document created: 2002

Document edited: 18.5.2011

2.1 Introduction

In solid and liquid materials heat transfer and viscous fluid flow are governed by heat and Navier-Stokes equations, which can be derived from the basic principles of conservation of mass, momentum and energy. Fluid can be either Newtonian or non-Newtonian. In the latter case the consideration in Elmer is limited to purely viscous behaviour with the power-law model.

In the following we present the governing equations of fluid flow, heat transfer and stresses in elastic material applied in Elmer. Also the most usual boundary conditions applied in computations are described.

2.2 Theory

The momentum and continuity equations can be written as

$$\rho \left(\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla) \vec{u} \right) - \nabla \cdot \bar{\bar{\sigma}} = \rho \vec{f}, \quad (2.1)$$

and

$$\left(\frac{\partial \rho}{\partial t} + (\vec{u} \cdot \nabla) \rho \right) + \rho (\nabla \cdot \vec{u}) = 0, \quad (2.2)$$

where $\bar{\bar{\sigma}}$ is the stress tensor. For Newtonian fluids

$$\bar{\bar{\sigma}} = 2\mu \bar{\bar{\varepsilon}} - \frac{2}{3}\mu (\nabla \cdot \vec{u}) \bar{\bar{I}} - p \bar{\bar{I}}, \quad (2.3)$$

where μ is the viscosity, p is the pressure, $\bar{\bar{I}}$ the unit tensor and $\bar{\bar{\varepsilon}}$ the linearized strain rate tensor, i.e.

$$\varepsilon_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right). \quad (2.4)$$

The density of an ideal gas depends on the pressure and temperature through the equation of state

$$\rho = \frac{p}{RT}, \quad (2.5)$$

where R is the gas constant:

$$R = \frac{\gamma - 1}{\gamma} c_p. \quad (2.6)$$

The specific heat ratio γ is defined as

$$\gamma = \frac{c_p}{c_v}, \quad (2.7)$$

where c_p and c_v are the heat capacities in constant pressure and volume, respectively. The value of γ depends solely on the internal molecular properties of the gas.

An incompressible flow is characterized by the condition $\rho = \text{constant}$, from which it follows that

$$\nabla \cdot \vec{u} = 0. \quad (2.8)$$

Enforcing the constraint (2.8) in (2.1), (2.2) and (2.3), the equations reduce to the Navier-Stokes equations

$$\rho \left(\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla) \vec{u} \right) - \nabla \cdot (2\mu \vec{\varepsilon}) + \nabla p = \rho \vec{f}, \quad (2.9)$$

$$\nabla \cdot \vec{u} = 0. \quad (2.10)$$

Compressible flows are modelled by the equations (2.1)-(2.7). Then, it is possible to replace the state equation (2.5) by

$$\rho = \frac{1}{c^2} p, \quad (2.11)$$

where $c = c(p, T, \dots)$ is the speed of sound. The equation (2.11) can be used with liquid materials as well.

Most commonly the term $\rho \vec{f}$ represents a force due to gravity, in which case the vector \vec{f} is the gravitational acceleration. It can also represent, for instance, the Lorentz force when magnetohydrodynamic effects are present.

For isothermal flows the equations (2.9) and (2.10) describe the system in full. For thermal flows also the heat equation needs to be solved.

For thermal incompressible fluid flows we assume that the Boussinesq approximation is valid. This means that the density of the fluid is constant except in the body force term where the density depends linearly on temperature through the equation

$$\rho = \rho_0 (1 - \beta(T - T_0)), \quad (2.12)$$

where β is the volume expansion coefficient and the subscript 0 refers to a reference state. Assuming that the gravitational acceleration \vec{g} is the only external force, then the force $\rho_0 \vec{g} (1 - \beta(T - T_0))$ is caused in the fluid by temperature variations. This phenomenon is called Grashof convection or natural convection.

One can choose between transient and steady state analysis. In transient analysis one has to set, besides boundary conditions, also initial values for the unknown variables.

2.2.1 Boundary Conditions

For the Navier-Stokes equation one can apply boundary conditions for velocity components or the tangential or normal stresses may be defined.

In 2D or axisymmetric cases the Dirichlet boundary condition for velocity component u_i is simply

$$u_i = u_i^b. \quad (2.13)$$

A value u_i^b can be constant or a function of time, position or other variables. In cylindrical cases the Dirichlet boundary condition for angular velocity u^θ is

$$u^\theta = \omega, \quad (2.14)$$

where ω is the rotation rate.

In axisymmetric geometries one has to set $u_r = 0$ and $\partial u_z / \partial r = 0$ on the symmetry axis. If there is no flow across the surface, then

$$\vec{u} \cdot \vec{n} = 0 \quad (2.15)$$

where \vec{n} is the outward unit normal to the boundary.

Surface stresses can be divided into normal and tangential stresses. Normal stress is usually written in the form

$$\sigma_n = \frac{\gamma}{R} - p_a \quad (2.16)$$

where γ is the surface tension coefficient, R the mean curvature and p_a the atmospheric (or external) pressure. Tangential stress has the form

$$\vec{\sigma}_\tau = \nabla_s \gamma, \quad (2.17)$$

where ∇_s is the surface gradient operator.

The coefficient γ is a thermophysical property depending on the temperature. Temperature differences on the surface influence the transport of momentum and heat near the surface. This phenomenon is called Marangoni convection or thermocapillary convection. The temperature dependence of the surface tension coefficient can be approximated by a linear relation:

$$\gamma = \gamma_0(1 - \vartheta(T - T_0)), \quad (2.18)$$

where ϑ is the temperature coefficient of the surface tension and the subscript 0 refers to a reference state. If a Boussinesq hypothesis is made, i.e., the surface tension coefficient is constant except in (2.17) due to (2.18), the boundary condition for tangential stress becomes

$$\vec{\sigma}_\tau = -\vartheta \gamma_0 \nabla_s T. \quad (2.19)$$

In equation (2.16) it holds then that $\gamma = \gamma_0$. The linear temperature dependence of the surface tension coefficient is naturally only one way to present the dependence. In fact, the coefficient γ can be any user defined function in Elmer. One may also give the force vector on a boundary directly as in

$$\vec{\sigma} \cdot \vec{n} = \vec{g}. \quad (2.20)$$

2.2.2 Linearization

As is well known, the convective transport term of the Navier-Stokes equations and the heat equation is a source of both physical and numerical instability. The numerical instability must be compensated somehow in order to solve the equations on a computer. For this reason the so called stabilized finite element method ([2],[1]) is used in Elmer to discretize these equations.

The convection term of the Navier-Stokes equations is nonlinear and has to be linearized for computer solution. There are two linearizations of the convection term in Elmer:

$$(\vec{u} \cdot \nabla) \vec{u} \approx (\vec{u} \cdot \nabla) \vec{u} \quad (2.21)$$

and

$$(\vec{u} \cdot \nabla) \vec{u} \approx (\vec{u} \cdot \nabla) \vec{u} + (\vec{u} \cdot \nabla) \vec{u} - (\vec{u} \cdot \nabla) \vec{u}, \quad (2.22)$$

where \vec{u} is the velocity vector from the previous iteration. The first of the methods is called Picard iteration or the method of the fixed point, while the latter is called Newton iteration. The convergence rate of the Picard iteration is of first order, and the convergence might at times be very slow. The convergence rate of the Newton method is of second order, but to successfully use this method, a good initial guess for velocity and pressure fields is required. The solution to this problem is to first take a couple of Picard iterations, and switch to Newton iteration after the convergence has begun.

2.2.3 Arbitrary Lagrangian-Eulerian (ALE) coordinates

For problems involving deformations the transient Navier-Stokes equation must be solved using Arbitrary Lagrangian-Eulerian (ALE) frame of reference. Assume that the mesh velocity during the nonlinear iteration is \vec{c} . Then the convective term yields

$$((\vec{u} - \vec{c}) \cdot \nabla) \vec{u} \approx ((\vec{U} - \vec{c}) \cdot \nabla) \vec{u}. \quad (2.23)$$

This results naturally to Picard iteration. For Newton iteration the additional two terms remains the same since the mesh velocities in there cancel each other.

2.2.4 Non-newtonian Material Models

There are several non-newtonian material models. All are functions of the strainrate $\dot{\gamma}$. The simple power law model has a problematic behavior at low shear rates. The more complicated models provide a smooth transition from low to high shearrates.

Power law

$$\eta = \begin{cases} \eta_0 \dot{\gamma}^{n-1} & \text{if } \dot{\gamma} > \dot{\gamma}_0, \\ \eta_0 \dot{\gamma}_0^{n-1} & \text{if } \dot{\gamma} \leq \dot{\gamma}_0. \end{cases} \quad (2.24)$$

where η_∞ is constant, $\dot{\gamma}_0$ is the critical shear rate, and n is the viscosity exponent.

Carreau-Yasuda

$$\eta = \eta_\infty + \Delta\eta (1 + (c\dot{\gamma})^y)^{\frac{n-1}{y}}, \quad (2.25)$$

where η_∞ is the high shearrate viscosity $\dot{\gamma} \rightarrow \infty$ provided that $n < 1$. For shearrates approaching zero the viscosity is $\eta_0 = \eta_\infty + \Delta\eta$. $\Delta\eta$ is thus the maximum viscosity difference between low and high shearrate. This model recovers the plain Carreau model when the Yasuda exponent $y = 2$.

Cross

$$\eta = \eta_\infty + \frac{\Delta\eta}{1 + c\dot{\gamma}^n}, \quad (2.26)$$

where again η_∞ is the high shearrate viscosity.

Powell-Eyring

$$\eta = \eta_\infty + \Delta\eta \frac{\text{asinh}(c\dot{\gamma})}{c\dot{\gamma}}. \quad (2.27)$$

All the viscosity models can be made temperature dependent. The current choice is to multiply the suggested viscosity with a factor $\exp(d(1/(T_o + T) - 1/T_r))$, where d is the exponential factor, T_o is temperature offset (to allow using of Celcius), and T_r the reference temperature for which the factor becomes one.

2.2.5 Flow in Porous Media

A simple porous media model is provided in the Navier-Stokes solver. It utilizes the Darcy's law that states that the flow resistance is propotional to the velocity and thus the modified momentum equation reads

$$\rho \left(\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla) \vec{u} \right) - \nabla \cdot \vec{\sigma} + r\vec{u} = \rho \vec{f}, \quad (2.28)$$

where r is the porous resistivity which may also be an orthotropic tensor. Usually the given parameter is permeability which is the inverse of the resistivity as defined here. No other features of the porous media flow is taken into consideration. Note that for large value of r only the bubble stabilization is found to work.

2.2.6 Rotating coordinates

In rotating coordinate system around origin one may define the angular velocity vector, $\vec{\Omega}$. The rotation introduces additional forces that may be evaluated from the following

$$\frac{d\vec{u}_{inertial}}{dt} = \frac{d\vec{u}_{rotating}}{dt} + 2\vec{\Omega} \times \vec{u}_{rotating} + \vec{\Omega} \times (\vec{\Omega} \times \vec{x}). \quad (2.29)$$

In numerical implementation the following Lagrange's formula is used

$$\vec{\Omega} \times (\vec{\Omega} \times \vec{x}) = (\vec{\Omega} \cdot \vec{x})\vec{\Omega} - (\vec{\Omega} \cdot \vec{\Omega})\vec{x}. \quad (2.30)$$

which results to the following form of the Navier-Stokes equation in rotating coordinates

$$\rho \left(\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla) \vec{u} \right) - \nabla \cdot \vec{\sigma} + 2\rho \vec{\Omega} \times \vec{u} = \rho(\vec{\Omega} \cdot \vec{\Omega})\vec{x} - \rho(\vec{\Omega} \cdot \vec{x})\vec{\Omega} + \rho \vec{f}, \quad (2.31)$$

It should be noted that now also the boundary conditions need to be given in the rotational coordinate system.

2.2.7 Coupling to Electric Fields

In electrokinetics the fluid may have charges that are coupled to external electric fields. This results to an external force that is of the form

$$\vec{f}_e = -\rho_e \nabla \phi, \quad (2.32)$$

where ρ_e is the charge density and ϕ is the external electric field. The charge density may also be a variable. More specifically this force may be used to couple the Navier-Stokes equation to the Poisson-Boltzmann equation describing the charge distribution in electric doubly layers. Also other types of forces that are proportional to the gradient of the field may be considered.

2.2.8 Coupling to Magnetic Fields

If the fluid has free charges it may couple with an magnetic field. The magnetic field induced force term for the flow momentum equations is defined as

$$\vec{f}_m = \vec{J} \times \vec{B}, \quad (2.33)$$

Here \vec{B} and \vec{E} are the magnetic and electric fields, respectively. The current density \vec{J} is defined as

$$\vec{J} = \sigma(\vec{E} + \vec{u} \times \vec{B}). \quad (2.34)$$

2.3 Keywords

Constants

Gravity Size 4 Real [x y z abs]

The above statement gives a real vector whose length is four. In this case the first three components give the direction vector of the gravity and the fourth component gives its intensity.

Solver solver id

Note that all the keywords related to linear solver (starting with Linear System) may be used in this solver as well. They are defined elsewhere.

Equation String [Navier-Stokes]

The name of the equation.

Flow Model String [Full][No convection][Stokes]

Flow model to be used. The default is to include both convection and time derivative terms in the model. The "No convection" model switches off the convection terms, and the "Stokes" model both the convection terms and the (explicit) time derivative terms.

Nonlinear System Convergence Tolerance Real

this keyword gives a criterion to terminate the nonlinear iteration after the relative change of the norm of the field variable between two consecutive iterations is small enough

$$||u_i - u_{i-1}|| < \epsilon ||u_i||,$$

where ϵ is the value given with this keyword.

Nonlinear System Max Iterations Integer

The maximum number of nonlinear iterations the solver is allowed to do.

Nonlinear System Newton After Iterations Integer

Change the nonlinear solver type to Newton iteration after a number of Picard iterations have been performed. If a given convergence tolerance between two iterations is met before the iteration count is met, it will switch the iteration type instead.

Nonlinear System Newton After Tolerance Real

Change the nonlinear solver type to Newton iteration, if the relative change of the norm of the field variable meets a tolerance criterion:

$$||u_i - u_{i-1}|| < \epsilon ||u_i||,$$

where ϵ is the value given with this keyword.

Nonlinear System Relaxation Factor Real

Giving this keyword triggers the use of relaxation in the nonlinear equation solver. Using a factor below unity is sometimes required to achieve convergence of the nonlinear system. A factor above unity might speed up the convergence. Relaxed variable is defined as follows:

$$u'_i = \lambda u_i + (1 - \lambda) u_{i-1},$$

where λ is the factor given with this keyword. The default value for the relaxation factor is unity.

Steady State Convergence Tolerance Real

With this keyword a equation specific steady state or coupled system convergence tolerance is given. All the active equation solvers must meet their own tolerances before the whole system is deemed converged. The tolerance criterion is:

$$||u_i - u_{i-1}|| < \epsilon ||u_i||,$$

where ϵ is the value given with this keyword.

Stabilize Logical

If this flag is set true the solver will use stabilized finite element method when solving the Navier-Stokes equations. Usually stabilization of the equations must be done in order to successfully solve the equations. If solving for the compressible Navier-Stokes equations, a bubble function formulation is used instead of the stabilized formulation regardless of the setting of this keyword. Also for the incompressible Navier-Stokes equations, the bubbles may be selected by setting this flag to False.

Div Discretization Logical

In the case of incompressible flow using the this form of discretization of the equation may lead to more stable discretization when the Reynolds number increases.

Gradp Discretization Logical

Whit this form of discretization pressure Dirichlet boundary conditions can be used (and pressure level must be fixed by such a condition). Also the mass flux is available as a natural boundary condition.

Equation eq id

The equation section is used to define a set of equations for a body or set of bodies:

Navier-Stokes Logical
if set to True, solve the Navier-Stokes equations.

Magnetic Induction Logical
If set to True, solve the magnetic induction equation along with the Navier-Stokes equations.

Convection String [None, Computed, Constant]
The convection type to be used in the heat equation, one of: None, Computed, Constant.
The second choice is used for thermal flows.

Body Force bf id
The body force section may be used to give additional force terms for the equations.

Boussinesq Logical
If set true, sets the Boussinesq model on.

Flow BodyForce i Real
May be used to give additional body force for the flow momentum equations, $i=1, 2, 3$.

Lorentz Force Logical
If set true, triggers the magnetic field force for the flow momentum equations.

Potential Force Logical
If this is set true the force used for the electricstatic coupling is activated.

Potential Field Real
The field to which gradient the external force is proportional to. For example the electrostatic field.

Potential Coefficient Real
The coefficient that multiplies the gradient term. For example, the charge density.

Angular Velocity Real
The angular velocity $\vec{\Omega}$ used for rotating coordinate systems. The size is always expected to be three.

Initial Condition ic id
The initial condition section may be used to set initial values for the field variables. The following variables are active:

Pressure Real

Velocity i Real
For each velocity component $i=1, 2, 3$.

Kinetic Energy Real
For the $k-\epsilon$ turbulence model.

Kinetic Energy Dissipation Real

Material mat id
The material section is used to give the material parameter values. The following material parameters may be set in Navier-Stokes equation.

Density Real The value of density is given with this keyword. The value may be constant, or variable. For the of compressible flow, the density is computed internally, and this keyword has no effect.

Viscosity Real
The relationship between stress and strain velocity. When using the solidification modelling, a viscosity-temperature curve must be given. The viscosity must be set to high enough value in the temperature range for solid material to effectively set the velocity to zero.

Reference Temperature Real
This is the reference temperature for the Boussinesq model of temperature dependence of density.

Heat Expansion Coefficient `real`

For the Boussinesq model the heat expansion coefficient must be given with this keyword. Default is 0.0.

Applied Magnetic Field `i Real`

An applied magnetic field may be given with these keywords with $i=1, 2, 3$.

Compressibility Model `String`

This setting may be used to set the compressibility model for the flow simulations. Currently the setting may be set to either `Incompressible`, `Perfect Gas` and `ArtificialCompressible`. If perfect gas model is chosen the settings `Reference Pressure` and `Specific Heat Ratio` must also be given. The artificial compressibility model may be used to boost convergence in fluid-structure-interaction cases. The default value of this setting is `Incompressible`.

Reference Pressure `Real`

with this keyword a reference level of pressure may be given. This setting applies only if the `Compressibility Model` is set to the value `Perfect Gas`.

Specific Heat Ratio `Real`

The ratio of specific heats (in constant pressure versus in constant volume) may be given with this keyword. This setting applies only if the `Compressibility Model` is set to value `Perfect Gas`. The default value of this setting is $5/3$, which is the appropriate value for monoatomic ideal gas.

For the $k-\varepsilon$ turbulence model the model parameters may also be given in the material section using the following keywords

KE `SigmaK Real [1.0]`

KE `SigmaE Real [1.3]`

KE `C1 Real [1.44]`

KE `C2 Real [1.92]`

KE `Cmu Real [0.09]`

Non-newtonian material laws are also defined in material section. For the power law the constant coefficient is given by the keyword `Viscosity`.

Viscosity Model `String`

The choices are `power law`, `carreau`, `cross`, `powell eyring` and `thermal carreau`. If none is given the fluid is treated as newtonian.

Viscosity Exponent `Real`

Parameter n in the models `power law`, `Carreau`, `Cross`

Viscosity Difference `Real`

Difference $\Delta\eta$ between high and low shear rate viscosities. Applicable to `Carreau`, `Cross` and `Powell-Eyring` models.

Viscosity Transition `Real`

Parameter c in the `Carreau`, `Cross` and `Powell-Eyring` models.

Critical Shear Rate `Real [0.0]`

Optional parameter $\dot{\gamma}_0$ in power law viscosity model.

Nominal Shear Rate `Real [0.0]`

Optional parameter in the power law viscosity model that gives the shear rate that returns the plain newtonian viscosity.

Yasuda Exponent `Real`

Optional parameter y in `Carreau` model. The default is 2. If activated the model is the more generic `Yasuda-Carreau` model.

Viscosity Temp Offset Real

Parameter T_o in the thermal viscosity dependence. When using Celcius instead of Kelvins this would be 273.15, for example.

Viscosity Temp Ref Real

Parameter T_r in the thermal viscosity dependence. This should be set so that unity factor is obtained when $T_r = T_o + T$.

Viscosity Temp Exp Real

Exponential parameter d in the thermal viscosity dependence.

Porosity is defined by the material properties

Porous Media Logical

If this keyword is set True then the porous model will be active in the material.

Porous Resistance Real

This keyword may give a constant resistance or also a orthotropic resistance where the resistance of each velocity component is given separately.

Boundary Condition bc id

The boundary condition section holds the parameter values for various boundary condition types. Dirichlet boundary conditions may be set for all the primary field variables. The one related to Navier-Stokes equation are

Velocity i Real

Dirichlet boundary condition for each velocity component $i = 1, 2, 3$.

Pressure Real

Absolute pressure.

Normal-Tangential Velocity Real

The Dirichlet conditions for the vector variables may be given in normal-tangential coordinate system instead of the coordinate axis directed system using the keywords

Flow Force BC Logical

Set to true, if there is a force boundary condition for the Navier-Stokes equations.

Surface Tension Expansion Coefficient Real

Triggers a tangential stress boundary condition to be used. If the keyword Surface Tension Expansion Coefficient is given, a linear dependence of the surface tension coefficient on the temperature is assumed. Note that this boundary condition is the tangential derivative of the surface tension coefficient

Surface Tension Coefficient Real

Triggers the same physical model as the previous one except no linearity is assumed. The value is assumed to hold the dependence explicitly.

External Pressure Real

A pressure boundary condition directed normal to the surface.

Pressure i Real

A pressure force in the given direction $i = 1, 2, 3$.

Free Surface Logical

Specifies a free surface.

Free Moving Logical

Specifies whether the regeneration of mesh is free to move the nodes of a given boundary when remeshing after moving the free surface nodal points. The default is that the boundary nodes are fixed.

The k- ε turbulence model also has its own set of boundary condition keywords (in addition to the Dirichlet settings):

Wall Law Logical

The flag activates the (Reichardt's) law of the wall for the boundary specified. the default is 9.0.

Boundary Layer Thickness Real

The distance from the boundary node of the meshed domain to the physical wall.

Bibliography

- [1] L.P. Franca and S.L. Frey. *Computer methods in Applied Mechanics and Engineering*, 99:209–233, 1992.
- [2] L.P. Franca, S.L. Frey, and T.J.R. Hughes. *Computer methods in Applied Mechanics and Engineering*, 95:253–276, 1992.

Model 3

Advection-Diffusion Equation

Module name: AdvectionDiffusion

Module subroutines: AdvectionDiffusionSolver

Module authors: Juha Ruokolainen, Ville Savolainen, Antti Pursula

Document authors: Ville Savolainen, Antti Pursula

Document edited: Oct 29th 2003

3.1 Introduction

Advection-diffusion equation (sometimes called diffusion-convection equation) describes the transport of a scalar quantity or a chemical species by convection and diffusion. The difference in the nomenclature usually indicates that an advected quantity does not have an effect on the velocity field of the total fluid flow but a convected quantity has. Advection-diffusion equation is derived from the principle of mass conservation of each species in the fluid mixture. Advection-diffusion equation may have sources or sinks, and several advection-diffusion equations may be coupled together via chemical reactions.

Fick's law is used to model the diffusive flux. Diffusion may be anisotropic, which may be physically reasonable at least in solids. If the velocity field is identically zero, the advection-diffusion equation reduces to the diffusion equation, which is applicable in solids.

Heat equation is a special case of the advection-diffusion (or diffusion-convection) equation, and it is described elsewhere in this manual.

3.2 Theory

3.2.1 Governing Equations

The advection-diffusion equation may, in general, be expressed in terms of relative or absolute mass or molar concentrations. In Elmer, when the transported quantity is carried by an incompressible fluid (or it is diffused in a solid), relative mass concentration $c_i = C_i/\rho$ for the species i is used (C_i is the absolute mass concentration in units kg/m^3 , and ρ the total density of the mixture). We have used the approximation valid for dilute multispecies flows, i.e., $0 \leq c_i \ll 1$. The advection-diffusion equation is now written as

$$\rho \left(\frac{\partial c_i}{\partial t} + (\vec{v} \cdot \nabla) c_i \right) = \rho \nabla \cdot (D_i \nabla c_i) + S_i, \quad (3.1)$$

where \vec{v} is the advection velocity, D_i the diffusion coefficient and S_i is a source, sink or a reaction term. The diffusion coefficient may be a tensor.

For a compressible fluid, the concentration should be expressed in absolute mass units, and the advection-diffusion equation reads

$$\frac{\partial C_i}{\partial t} + (\nabla \cdot \vec{v}) C_i + (\vec{v} \cdot \nabla) C_i = \nabla \cdot (D_i \nabla C_i) + S_i. \quad (3.2)$$

For a situation, where the quantity is transported through a phase change boundary, it is convenient to scale the absolute mass formulation by the respective solubilities of the different phases. Such a case is for example the surface of a liquid, where the transported quantity is evaporated into a gaseous material. The scaled concentration variable satisfies the equilibrium boundary condition on the phase change boundary automatically, and thus the advection-diffusion equation can be solved for both materials simultaneously. The scaling is following

$$x_i = \frac{C_i}{C_{i,max}}, \quad (3.3)$$

where x_i is the concentration of species i relative to its maximum solubility in the current material in absolute mass units. The maximum solubility has to be a constant (temperature independent) for the absolute mass formulation of the advection-diffusion equation to remain unchanged.

It is also possible to include temperature dependent diffusion (Soret diffusion). This introduces an additional term on the right hand side of the equation:

$$\nabla \cdot (\rho D_{i,T} \nabla T), \quad (3.4)$$

where $D_{i,T}$ is the thermal diffusion coefficient of species i . The coefficient $D_{i,T}$ has to be given in the units m^2/Ks regardless of the units used for concentration.

The velocity of the advecting fluid, \vec{v} , is typically calculated by the Navier-Stokes equation and read in from a restart file. All quantities can also be functions of, e.g., temperature that is given or solved by the heat equation. Several advection-diffusion equations for different species i may be coupled and solved for the same velocity field.

Given volume species sources S_i can be prescribed. They are given in absolute mass units, i.e., kg/m^3s . If the equation is scaled to maximum solubility, the source term can be given in absolute mass units, or in scaled units, $S_{i,sc} = S_i/C_{i,max}$, which is the default.

3.2.2 Boundary Conditions

For each species one can apply either a prescribed concentration or a mass flux as boundary conditions.

Dirichlet boundary condition reads as

$$c_i = c_{i,b}, \quad (3.5)$$

or

$$C_i = C_{i,b}, \quad (3.6)$$

depending on the units. If the concentration is scaled to maximum solubility, the Dirichlet boundary conditions have to be given also in scaled values, $x_i = C_{i,b}/C_{i,max}$. In all variations, the boundary value can be constant or a function of time, position or other variables.

One may specify a mass flux \vec{j}_i perpendicular to the boundary by

$$\vec{j}_i \cdot \vec{n} = -D_i \frac{\partial C_i}{\partial n} = g. \quad (3.7)$$

In relative mass units, this may be written as

$$\vec{j}_i \cdot \vec{n} = -\rho D_i \frac{\partial c_i}{\partial n} = g. \quad (3.8)$$

Thus the units in the flux boundary condition are always kg/m^2s except when the equation is scaled to maximum solubility. In that case the default is to give flux condition in scaled units, $g_{sc} = g/C_{i,max}$, although the physical units are also possible.

The mass flux may also be specified by a mass transfer coefficient β and an external concentration C_{ext}

$$-D_i \frac{\partial C_i}{\partial n} = \beta(C_i - C_{i,ext}). \quad (3.9)$$

On the boundaries where no boundary condition is specified, the boundary condition $g = 0$ is applied. This zero flux condition is also used at a symmetry axis in 2D, axisymmetric or cylindrical problems.

The equilibrium boundary condition on phase change boundaries under certain conditions is that the relative amounts of the transported quantity are equal on both sides of the boundary,

$$\frac{C_i^{(1)}}{C_{i,max}^{(1)}} = \frac{C_i^{(2)}}{C_{i,max}^{(2)}}, \quad (3.10)$$

where the superscripts (1) and (2) refer to different sides of the boundary. This boundary condition is automatically satisfied if the equation is scaled with the maximum solubilities $C_{i,max}^{(j)}$.

However, the scaling causes a discontinuity into the mass flux of the species through the phase change surface. The solver compensates this effect as long as such a boundary is flagged in the command file by the user.

3.3 Keywords

Simulation

The simulation section gives the case control data:

Simulation Type String

Advection-diffusion equation may be either Transient or Steady State.

Coordinate System String

Defines the coordinate system to be used, one of: Cartesian 1D, Cartesian 2D, Cartesian 3D, Polar 2D, Polar 3D, Cylindric, Cylindric Symmetric and Axi Symmetric.

Timestepping Method String

Possible values of this parameter are Newmark (an additional parameter Newmark Beta must be given), BDF (BDF Order must be given). Also as a shortcut to Newmark-method with values of Beta=0.0, 0.5, 1.0 the keywords Explicit Euler, Crank-Nicolson, and Implicit Euler may be given respectively. The recommended choice for the first order time integration is the BDF method of order 2.

BDF Order Integer

Value may range from 1 to 5.

Newmark Beta Real

Value in range from 0.0 to 1.0. The value 0.0 equals to the explicit Euler integration method and the value 1.0 equals to the implicit Euler method.

Solver solver id

The solver section defines equation solver control variables. Most of the possible keywords – related to linear algebra, for example – are common for all the solvers and are explained elsewhere.

Equation String [Advection Diffusion Equation Varname]

The name of the equation, e.g., Advection Diffusion Equation Oxygen.

Variable String Varname

The name of the variable, e.g., Oxygen.

Procedure File "AdvectionDiffusion" "AdvectionDiffusionSolver"

The name of the file and subroutine.

Nonlinear System Convergence Tolerance Real

The criterion to terminate the nonlinear iteration after the relative change of the norm of the field variable between two consecutive iterations k is small enough

$$\|u_k - u_{k-1}\| < \epsilon \|u_k\|,$$

where ϵ is the value given with this keyword, and u is either c_i or C_i .

Nonlinear System Max Iterations Integer

The maximum number of nonlinear iterations the solver is allowed to do.

Nonlinear System Relaxation Factor Real

Giving this keyword triggers the use of relaxation in the nonlinear equation solver. Using a factor below unity is sometimes required to achieve convergence of the nonlinear system. A factor above unity might speed up the convergence. Relaxed variable is defined as follows:

$$u'_k = \lambda u_k + (1 - \lambda)u_{k-1},$$

where λ is the factor given with this keyword. The default value for the relaxation factor is unity.

Steady State Convergence Tolerance Real

With this keyword a equation specific steady state or coupled system convergence tolerance is given. All the active equation solvers must meet their own tolerances for their variable u before the whole system is deemed converged. The tolerance criterion is:

$$||u_i - u_{i-1}|| < \epsilon ||T_i||,$$

where ϵ is the value given with this keyword.

Stabilize Logical

If this flag is set true the solver will use stabilized finite element method when solving the advection-diffusion equation with a convection term. If this flag is set to False, RFB (Residual Free Bubble) stabilization is used instead (unless the next flag Bubbles is set to False in a problem with Cartesian coordinate system). If convection dominates, some form of stabilization must be used in order to successfully solve the equation. The default value is False.

Bubbles Logical

There is also a residual-free-bubbles formulation of the stabilized finite-element method. It is more accurate and does not include any ad hoc terms. However, it may be computationally more expensive. The default value is True. If both Stabilize and Bubbles are set to False, no stabilization is used. This choice may be enforced in a problem with Cartesian coordinates, but the results might be nonsensical. Both Stabilize and Bubbles should not be set to True simultaneously.

Equation eq id

The equation section is used to define a set of equations for a body or set of bodies.

Advection Diffusion Equation Varname Logical

If set to True, solve the advection-diffusion equation.

Convection String

The type of convection to be used in the advection-diffusion equation, one of: None, Computed, Constant.

Concentration Units String

If set to Absolute Mass, absolute mass units are used for concentration. Recommended for a compressible flow. Also possible to select Mass To Max Solubility which causes the absolute mass formulation of the equation to be scaled by the maximum solubilities of each material.

Body Forces bf id

The body force section may be used to give additional force terms for the equations. The following keyword is recognized by the solver:

Varname Diffusion Source Real

An additional volume source for the advection-diffusion equation may be given with this keyword. It may depend on coordinates, temperature and other variables, such as concentration of other chemical species, and thus describe a source, a sink or a reaction term. Given in absolute mass units or, in case of scaling, in the scaled units.

Physical Units Logical True

With this keyword, the source term can be given in absolute mass units regardless of scaling.

Initial Condition ic id

The initial condition section may be used to set initial values for the concentration c_i , C_i or x_i .

Varname Real

Material mat id

The material section is used to give the material parameter values. The following material parameters may be effective when advection-diffusion equation is solved.

Convection Velocity i Real

Convection velocity $i=1, 2, 3$ for the constant convection model.

Density Real

The value of density of the transporting fluid is given with this keyword. The value may be constant, or variable. For compressible flow, the density of the transporting fluid is computed internally, and this keyword has no effect.

Compressibility Model String

This setting may be used to set the compressibility model for the flow simulations. Choices are Incompressible and Perfect Gas. If set to the latter, the density is calculated from the ideal gas law. Then also the settings Reference Pressure, Specific Heat Ratio and Heat Capacity must be given.

Reference Pressure Real

With this keyword a reference level of pressure may be given.

Specific Heat Ratio Real

The ratio of specific heats (in constant pressure versus in constant volume) may be given with this keyword. The default value of this setting is $5/3$, which is the appropriate value for monoatomic ideal gas.

Heat Capacity Real

For the compressible flow, specific heat in constant volume.

Varname Diffusivity Real

The diffusivity D given by, e.g., Oxygen Diffusivity. Can be a constant or variable. For an anisotropic case, may also be a tensor D_{ij} .

Varname Soret Diffusivity Real

The thermal diffusivity coefficient D_T given by, e.g., Oxygen Soret Diffusivity. Can be a constant or variable.

Varname Maximum Solubility Real

The maximum solubility of the species in absolute mass units. Has to be a constant value.

Boundary Condition bc id

In advection-diffusion equation we may set the concentration directly by Dirichlet boundary conditions or use mass flux condition. The natural boundary condition is zero flux condition.

Varname Real

Mass Transfer Coefficient Real

External Concentration Real

These two keywords are used to define flux condition that depends on the external concentration and a mass transfer coefficient. This condition is only applicable to absolute mass formulation of the equation (see keywords for Equation block).

Varname Flux Real

A user defined mass flux term in absolute mass units or, in case of scaling, in the scaled units.

Physical Units Logical True

With this keyword, the flux boundary condition can be given in absolute mass units regardless of scaling. Note that this keyword does NOT affect the Dirichlet boundary condition nor the mass transfer coefficient bc.

Varname Solubility Change Boundary Logical True

This keyword marks the boundary over which the maximum solubility changes. Has to be present for the mass flux continuity to be preserved.

Normal Target Body Integer bd id

In a solubility change boundary, this keyword can be used to control on which side the mass flux compensation is done. Basically, this can be done on either side but there can be some effect on the accuracy or on the speed of the solution. Recommended is to give as normal target the body with less dense mesh, or the direction of average species transport. If normal target body is not specified, the material with smaller density is used.

Model 4

Advection-Reaction Equation

Module name: AdvectionReaction

Module subroutines: AdvectionReactionSolver

Module authors: Mikko Lyly, Juha Ruokolainen, Thomas Zwinger

Document authors: Thomas Zwinger

Document edited: March 3rd 2009

4.1 Introduction

Advection-reaction equation describes the transport of a passive scalar quantity, c , by a fluid. The advected quantity is assumed not to have an effect on the velocity field. Besides a reaction rate, advection-reaction equation may have sources or sinks. If no reaction rate and source are given, this equation may be used to trace passive scalars through a given flow-field. If a constant source of unity value is given, the equation also may be used to evaluate the time a passive tracer has remained in the flow field.

4.2 Theory

4.2.1 Governing Equations

The advective transport of a scalar c can be written as

$$\frac{\partial c}{\partial t} + \vec{v} \cdot \nabla c + \Gamma c = S, \quad (4.1)$$

where \vec{v} is the advection velocity, Γ is the reaction rate and S is a source/sink, depending on the sign.

Due to the absence of any diffusion, (4.1) has to be solved applying the Discontinuous Galerkin (DG) method. Elmer implements the particular method as presented in [1]. In order to evaluate jumps across partition boundaries in parallel computations, DG implies the utilization of halo-elements for domain decomposition (see ElmerGrid manual for details).

4.2.2 Limiters

If the scalar has a lower, $c_{\min} \leq c$ and/or an upper limit $c \leq c_{\max}$ limit (where the limit can be also a function of another variable), the variational form of (4.1) becomes a variational inequality. In order to obtain a consistent solution a method using Dirichlet constraints within the domain is applied. The exact procedure is the following:

1. construct the linear system: $Ac = S$, with the system matrix A and the solution vector c on the left-hand side and the force vector S on the right hand side

2. set nodes as *active* if the constraint is violated
3. for *active* nodes the matrix and force vector are manipulated such that effectively a Dirichlet condition $c = c_{\max/\min}$ is applied
4. the manipulated system is solved: $\tilde{A}\tilde{c} = \tilde{S}$
5. a residual is obtained from the un-manipulated system: $R = A\tilde{c} - S$
6. an *active* node is reset if the residual is $R < 0$ (for lower limit) and $R > 0$ (for upper limit)

The whole algorithm is iterated (within the non-linear iteration loop) until the limit given in `Nonlinear System Convergence Tolerance` is reached. In the converged solution the residual represents the needed accumulation/volume flux (on matrix level, hence not in physical units) needed in order to obtain the limited solution. Consequently, the system not necessarily is volume conserving if the Dirichlet method is applied.

4.2.3 Boundary Conditions

At boundaries, a Dirichlet boundary condition reads as

$$c = c_b. \quad (4.2)$$

By nature of the applied DG method, the condition above only applies at inflow boundaries, i.e., if

$$\vec{v} \cdot \vec{n}_b < 0, \quad (4.3)$$

where \vec{n}_b is the outwards facing surface normal of the boundary.

On the boundaries where no boundary condition is specified, the boundary condition $c = 0$ is applied upon inflow.

4.3 Keywords

Simulation

The simulation section gives the case control data:

`Simulation Type String`

Advection-reaction equation may be either `Transient` or `Steady State`.

`Coordinate System String`

Defines the coordinate system to be used, one of: `Cartesian 1D`, `Cartesian 2D`, `Cartesian 3D`, `Polar 2D`, `Polar 3D`, `Cylindric`, `Cylindric Symmetric` and `Axi Symmetric`.

`Timestepping Method String`

Possible values of this parameter are `Newmark` (an additional parameter `Newmark Beta` must be given), `BDF` (`BDF Order` must be given). Also as a shortcut to `Newmark-method` with values of `Beta=0.0, 0.5, 1.0` the keywords `Explicit Euler`, `Crank-Nicolson`, and `Implicit Euler` may be given respectively. The recommended choice for the first order time integration is the `BDF` method of order 2.

`BDF Order Integer`

Value may range from 1 to 5.

`Newmark Beta Real`

Value in range from 0.0 to 1.0. The value 0.0 equals to the explicit Euler integration method and the value 1.0 equals to the implicit Euler method.

`Solver solver id`

The solver section defines equation solver control variables. Most of the possible keywords – related to linear algebra, for example – are common for all the solvers and are explained elsewhere.

Equation String [Advection Reaction Equation Variable_name]

The name of the equation, e.g., Advection Reaction Equation Tracer.

Discontinuous Galerkin Logical

needs to be set to true

Variable String Variable_name

The name of the variable, e.g., Tracer. As the variable is a DG variable (i.e., not renderable e.g. in ElmerPost), the user usually adds the option `-nooutput` in order to avoid output in the output files

Procedure File "AdvectionReaction" "AdvectionReactionSolver"

The name of the file and subroutine.

Nonlinear System Convergence Tolerance Real

The criterion to terminate the nonlinear iteration after the relative change of the norm of the field variable between two consecutive iterations k is small enough

$$\|c_k - c_{k-1}\| < \epsilon \|c_k\|,$$

where ϵ is the value given with this keyword.

Nonlinear System Max Iterations Integer

The maximum number of nonlinear iterations the solver is allowed to do.

Steady State Convergence Tolerance Real

With this keyword a equation specific steady state or coupled system convergence tolerance is given. All the active equation solvers must meet their own tolerances for their variable c before the whole system is deemed converged. The tolerance criterion is:

$$\|c_i - c_{i-1}\| < \epsilon \|c_i\|,$$

where ϵ is the value given with this keyword.

Limit Solution Logical

Assumes the variational inequality method to apply, if set to true.

Exported Variable 1 String

in order to write the DG variable Variable_name to a for ElmerPost (non-DG mesh) readable variable, an exported variable with an arbitrary name (e.g., Exported Variable 1 = Variable_name Nodal Result) has to be defined. It is then used to interpolate the DG result to nodal values in order to display them.

Equation eq id

The equation section is used to define a set of equations for a body or set of bodies.

Convection String

The type of convection to be used in the advection-reaction equation, one of: None, Computed, Constant.

Body Forces bf id

The body force section may be used to give additional force terms for the equations. The following keyword is recognized by the solver:

Variable_name Source Real

defines the volumetric source for variable c

Initial Condition ic id

The initial condition section may be used to set initial values for the scalar c .

Variable_name Real

Material mat id

The material section is used to give the material parameter values. The following material parameters may be effective when advection-diffusion equation is solved.

Convection Velocity i Real

Convection velocity $i=1, 2, 3$ for the constant convection model.

Variable_name Upper Limit Real

The upper limit, c_{\max} , for variable Variable_name. Only used if keyword Limit Solution for the solver is set to true

Variable_name Lower Limit Real

The lower limit, c_{\min} , for variable Variable_name. Only used if keyword Limit Solution for the solver is set to true

Variable_name Gamma Real

defines the reaction rate, Γ

Boundary Condition bc id

Variable_name Real sets the value for c at inflow boundaries

Bibliography

- [1] F. Brezzi and E. Marini, L. D. and Süli. Discontinuous Galerkin methods for first-order hyperbolic problems. *Math. Models Methods Appl. Sci.*, 14(12):1893–1903, 2004.

Model 5

Reynolds Equation for Thin Film Flow

Module name: ReynoldsSolver

Module subroutines: ReynoldsSolver, ReynoldsHeatingSolver

Module authors: Peter Råback

Module status: Alpha

Document authors: Peter Råback

Document created: 24.10.2007

Document edited: 24.10.2007

5.1 Introduction

The flow of fluids is in the continuum level usually described by the Navier-Stokes equations. For narrow channels this approach is an overkill and usually not even necessary. Neglecting the inertial forces and assuming fully developed laminar velocity profiles the flow equations may be reduced in dimension resulting to the Reynolds equation.

The current implementation of the Reynolds equation is suitable for incompressible and weakly compressible liquids as well as for isothermal and adiabatic ideal gases. The nonlinear terms for the compressible fluids are accounted for. The fluid is assumed to be newtonian i.e. there is a direct connection between the strain rate and stress. The equation may be solved either in steady state or in a transient mode.

There is an additional solver for postprocessing purposes that computes the local heat generation field using the Galerkin method. It also computes the integrals over the force and heating fields over the whole area.

5.2 Theory

The underlying assumption of the Reynolds equation is that the flow in the channel is fully developed and has thus the Hagen-Poiseuille parabolic velocity profile. Accounting also for the movement of the planes and leakage through perforation holes the pressure may be solved from the equation

$$\nabla \cdot \left(\frac{\rho h^3}{12\eta} \nabla p \right) - Y \rho p = \frac{1}{2} \nabla \cdot (\rho h \vec{v}_t) + h \frac{\partial \rho}{\partial t} + \rho v_n, \quad (5.1)$$

where ρ is the density, η is the viscosity, p is the pressure and h is the gap height, v_t is the tangential velocity, and v_n is the velocity in direction of the surface normal [1, 5]. Holes may be homogenized using the flow admittance Y which gives the ratio between pressure drop and mean flow velocity through the hole.

The exact form of the Reynolds equation depends on the material law for density, $\rho(p)$. The absolute value of density does not play any role and therefore we may study just the functional forms. For gases we solve for the pressure variation from the reference pressure P_0 rather than for the absolute pressure. The

different functional forms for some idealized material laws are the following:

$$\begin{aligned}\rho &\propto (P_0 + p) && \text{isothermal ideal gas} \\ \rho &\propto (P_0 + p)^{1/\gamma} && \text{adiabatic ideal gas} \\ \rho &\propto 1 && \text{incompressible} \\ \rho &\propto e^{p/\beta} && \text{weakly compressible.}\end{aligned}$$

Here $\gamma = C_p/C_V$ is the specific heat ratio and β the bulk modulus. In discretization of the equations it is also useful to derive the functional dependencies of the density derivatives in respect to pressure,

$$\begin{aligned}\rho_p &\propto 1 && \text{isothermal ideal gas} \\ \rho_p &\propto (1/\gamma)(P_0 + p)^{1/\gamma-1} && \text{adiabatic ideal gas} \\ \rho_p &\propto 0 && \text{incompressible} \\ \rho_p &\propto \rho/\beta && \text{weakly compressible.}\end{aligned}$$

In order to improve convergence of the iteration of the nonlinear system some terms including differentials of density may be expressed implicitly using pressure. This way equation (5.1) may be written in the following form:

$$\nabla \cdot \left(\frac{\rho h^3}{12\eta} \nabla p \right) - Y \rho p - \rho_p h \frac{\partial p}{\partial t} - \frac{1}{2} \rho_p h \vec{v}_t \cdot \nabla p = \frac{1}{2} \rho \nabla \cdot (h \vec{v}_t) + \rho v_n. \quad (5.2)$$

The surface velocity \vec{v} may also be given in normal cartesian coordinate system. Then the normal and tangential components may easily be obtained from

$$\begin{aligned}v_n &= \vec{v} \cdot \vec{n} \\ \vec{v}_t &= \vec{v} - v_n \vec{n}.\end{aligned}$$

The normal velocity and gap height are naturally related by

$$v_n = \frac{\partial h}{\partial t}. \quad (5.3)$$

In transient case the user should make sure that this relationship is honored.

5.2.1 Flow admittances of simple geometries

The flow admittance, Y , occurring in the Reynolds equation may sometimes be solved analytically for simple hole geometries from the steady-state Stokes equation. Generally Y depends on the history but here we assume that it presents the steady-state situation of the flow [2, 5]. This means that inertial and compressibility effects are not accounted for. For cylindrical holes the admittance then yields,

$$Y = \frac{D^2}{32\eta b}, \quad (5.4)$$

where D is the diameter of the holes and b is the length of the hole. In case of a narrow slot with width W the admittance is given by

$$Y = \frac{W^2}{12b\eta}. \quad (5.5)$$

5.2.2 Gas rarefaction effects

Generally the Reynolds equation could also be used to model nonnewtonian material laws. The current implementation is limited to the special case of rarefied gases. The goodness of the continuum assumption η depends on the Knudsen number, K_n , which is defined by

$$K_n = \frac{\lambda}{h}, \quad (5.6)$$

where λ is the mean free path of the molecules and h is the characteristic scale (here the gap height). In this solver only the dependence with pressure is taken into account from the formula

$$\lambda = \frac{1}{1 + p/P_0} \lambda_0. \quad (5.7)$$

When the Knudsen number is very small ($K_n \ll 1$) the gas may be considered as a continuous medium. When the Knudsen number is in the transition regime ($K_n \approx 1$) we may take the gas rarefaction effect into account by an effective viscosity. This accounts for the slip conditions of the flow in the channel by decreasing the viscosity value. An approximation given by Veijola [4] is

$$\eta = \frac{\eta_0}{1 + 9.638 K_n^{1.159}}. \quad (5.8)$$

Its relative accuracy is 5 % in the interval $0 < K_n < 880$.

5.2.3 Boundary conditions for the Reynolds equation

The Reynolds equation may have different boundary conditions. The natural boundary condition that is obtained by default is

$$\frac{\partial p}{\partial n} = 0. \quad (5.9)$$

This condition may be used at symmetry and closed boundaries.

If the aspect ratio of the resonator is large then the pressure variation at the open sides is small compared to the values far from boundaries. Then may set Dirichlet boundary conditions ($p = 0$) for the pressure. However, if the aspect ratio is relatively small the open side effects should be taken into account. The pressure variation at the side is not exactly zero while also the open space has a flow resistance. The pressure derivative at the boundary is approximated by

$$\frac{\partial p}{\partial n} = \frac{p}{L}, \quad (5.10)$$

where L is the effective added length of the open sides [3]. If gas rarefaction is not accounted for then $L = 0.8488h$, otherwise

$$L = 0.8488(1.0 + 2.676 K_n^{0.659})h. \quad (5.11)$$

5.2.4 Postprocessing

When the equation has been solved the solution may be used to compute some data for postprocessing purposes. The local volume flux in the lateral direction may be obtained from

$$\vec{q} = -\frac{h^3}{12\eta} \nabla p + h\vec{v}_t. \quad (5.12)$$

The total force acting on the surface is

$$\vec{F} = \int_A \left(p\vec{n} + \frac{\eta}{h} \vec{v}_t \right) dA, \quad (5.13)$$

where the first term is due to pressure driven flow and the second one due to sliding driven flow. Also the heating effect may be computed. It consist of two parts: pressure driven flow and sliding flow. The local form of this is

$$h = \frac{h^3}{12\eta} |\nabla p|^2 + \frac{\eta}{h} |\vec{v}_t|^2. \quad (5.14)$$

Therefore the total heating power of the system is

$$Q = \int_A q dA. \quad (5.15)$$

It should be noted that if the velocity field \vec{v} is constant then the integral quantities should fulfill the condition $Q = \vec{F} \cdot \vec{v}$.

Note that the above implementation does not take into account the leakage through perforation holes nor the compressibility effects of the fluids.

5.3 Keywords

The module includes two different solvers. `ReynoldsSolver` solves the differential equation (5.2) while `ReynoldsHeatingSolver` solves the equation (5.14) and computes the integrals. The second solver only makes sense when the pressure field has already been computed with the first one. The second solver uses the same material parameters as the first one.

Keywords for ReynoldsSolver

`Solver solver id`

`Equation String ReynoldsSolver`

A describing name for the solver. This can be changes as long as it is used consistently.

`Procedure File "ReynoldsSolver" "ReynoldsSolver"`

Name of the solver subroutine.

`Variable String FilmPressure`

The name of the variable may be freely chosen as far as it is used consistently also elsewhere.

`Variable DOFs Integer 1`

Degrees of freedom for the pressure. This should be 1 which is also the default value.

`Procedure File "ReynoldsSolver" "ReynoldsSolver"`

The name of the module and procedure. These are fixed.

`Apply Limiter Logical`

The generic soft limiters may be applied for the Reynolds equation in order to mimic the effects of cavitation. With this flag active the minimum and maximum limiters are accounted.

`Nonlinear System Convergence Tolerance Real`

The transient equation is nonlinear if the relative displacement or pressure deviation is high. The iteration is continued till the relative change in the norm falls under the value given by this keyword.

`Nonlinear System Max Iterations Integer`

This parameter gives the maximum number of nonlinear iterations required in the solution. This may be set higher than the typical number of iterations required as the iteration procedure should rather be controlled by the convergence tolerance.

`Material mat id`

`Gap Height Real`

Height of the gap where the fluid is trapped. If the case is transient the user should herself make sure that also this variable has the correct dependence on time.

`Surface Velocity i Real`

The velocity of the moving body may be given in either cartesian coordinates, or in ones that are already separated to normal and tangential directions. In the first case the velocity components are given with this keyword with $i=1, 2, 3$.

`Tangent Velocity i Real`

For setting the tangential velocity (i.e. sliding velocity) use this keyword with $i=1, 2, 3$.

`Normal Velocity Real`

Normal velocity is the velocity in the direction of the surface normal. Typically a negative value means contraction.

`Viscosity Real`

Viscosity of the gas.

`Viscosity Model String`

The choices are `newtonian` and `rarefied`. The first one is also the default.

Compressibility Model String

The choices are incompressible, weakly compressible, isothermal ideal gas, and adiabatic ideal gas.

Reference Pressure Real

Reference pressure is required only for the ideal gas laws.

Specific Heat Ratio Real

This parameter is only required for adiabatic processes. For ideal monoatomic gases the ratio is 5/3. Only required for the adiabatic compressibility model.

Bulk Modulus Real

The parameter β in the weakly compressible material model.

Mean Free Path Real

If the viscosity model assumes rarefied gases the mean free path of the gas molecules in the reference pressure must be given.

Flow Admittance Real

The steady-state flow admittance resulting from perforation, for example.

Body Force bf id

FilmPressure Lower Limit Real

The lower limit for the pressure that will be iteratively be enforced when the soft limiters are active.

Boundary Condition bc id

FilmPressure Real

Sets the boundary conditions for the pressure. Usually the deviation from reference pressure is zero at the boundaries.

Open Side Logical

The open end effect may be taken into account by setting this keyword True.

Keywords for ReynoldsPostprocess

This solver uses largely the same keywords that are already defined above. Only the Solver section has its own keyword settings. This solvers should be active in the same bodies than the ReynoldsSolver.

Solver solver id

Equation String ReynoldsPostprocess

A describing name for the solver. This can be changes as long as it is used consistently.

Procedure File "ReynoldsSolver" "ReynoldsSolver"

Name of the solver subroutine.

Reynolds Pressure Variable Name String

The name of the field that is assumed to provide the pressure field. The default is FilmPressure. Note that the Variable of this equation need not to be defined since it is automatically set when any of the field computation is requested.

Calculate Force Logical

Calculate the forces resulting from the pressure distribution computed with the Reynolds equation. The name of the field is obtained by adding the suffix Force.

Calculate Flux Logical

Calculate the fluxes resulting from the pressure distribution computed with the Reynolds equation. The name of the field is obtained by adding the suffix Flux.

Calculate Heating Logical

Calculate the heating efficiency from the pressure distribution computed with the Reynolds equation. The name of the field is obtained by adding the suffix Heating.

Calculate Force Dim Integer

By default the dimension of the force field is the mesh dimension plus one. Sometimes the pressure lives on a 1D line of a 2D mesh. Then this keyword may be used to suppress the dimension of force to two.

Calculate Flux Dim Integer

As the previous keyword but for the flux.

Bibliography

- [1] B.J. Hamrock, S.R. Schmid, and B.O. Jacobson. *Fundamentals of fluid film lubrication*. Marcel Dekker, second edition, 2004.
- [2] P. Rieback, A. Pursula, V. Junttila, and T. Veijola. Hierarchical finite element simulation of perforated plates with arbitrary hole geometries. In *NANOTECH 2003, San Francisco, 23-27 February 2003*, volume 1, pages 194–197.
- [3] T. Veijola. *APLAC 7.60 Reference Manual*, January 2002. Electromechanical Macro Models.
- [4] T. Veijola, H. Kuisma, J. Lahdenperä, and T. Rytönen. Equivalent-circuit model of the squeezed gas film in a silicon accelerometer. *Sensors and Actuators A*, pages 239–248, 1995.
- [5] T. Veijola and P. Rieback. A method for solving arbitrary mems perforation problems with rare gas effects. In *NANOTECH 2005, Anaheim, May 8-12, 2005*, volume 3, pages 561–564.

Part II

Models of Solid Mechanics

Model 6

Linear Elasticity

Module name: StressSolve

Module subroutines: StressSolver

Module authors: Juha Ruokolainen

Document authors: Juha Ruokolainen

Document edited: 22.04.2007

6.1 Introduction

This module computes displacement field from the Navier equations. The Navier equations correspond to linear theory of elastic deformation of solids. The material may be anisotropic and stresses may be computed as a post processing step, if requested by the user. Thermal stresses may also be requested.

6.2 Theory

The dynamical equation for elastic deformation of solids may be written as

$$\rho \frac{\partial^2 \vec{d}}{\partial t^2} - \nabla \cdot \tau = \vec{f}, \quad (6.1)$$

where ρ is density, \vec{d} is the displacement field, \vec{f} given volume force, and τ the stress tensor. Stress tensor is given by

$$\tau^{ij} = C^{ijkl} \varepsilon_{kl} - \beta^{ij} (T - T_0), \quad (6.2)$$

where ε is the strain and quantity C is the elastic modulus. The elastic modulus is a fourth order tensor, which has at the most 21 (in 3D, 10 in 2D) independent components due to symmetries. In Elmer thermal stresses may be considered by giving the heat expansion tensor β and reference temperature of the stress free state T_0 . The temperature field T may be solved by the heat equation solver or otherwise. The linearized strains are given simply as:

$$\varepsilon = \frac{1}{2} (\nabla \vec{d} + (\nabla \vec{d})^T). \quad (6.3)$$

6.2.1 Material laws

For isotropic materials the elastic modulus tensor may be reduced to two independent values, either the Lamé parameters, or equivalently to Young's modulus and Poisson ratio. The stress tensor given in terms of Lamé parameters is:

$$\tau = 2\mu\varepsilon + \lambda \nabla \cdot \vec{d} I - \beta (T - T_0) I, \quad (6.4)$$

where μ and λ are the first and second Lamé parameters respectively, β the heat expansion coefficient, and I is the unit tensor. Lamé parameters in terms of Young's modulus and Poisson ratio read

$$\lambda = \frac{Y\kappa}{(1+\kappa)(1-2\kappa)}, \quad \mu = \frac{Y}{2(1+\kappa)} \quad (6.5)$$

except for plane stress situations ($\tau_z = 0$) where λ is defined as

$$\lambda = \frac{Y\kappa}{(1-\kappa^2)}. \quad (6.6)$$

Quantities Y and κ are the Young's modulus and Poisson ratio respectively.

For anisotropic materials, the stress-strain relations may be given in somewhat different form:

$$\tau_V = E\varepsilon_V, \quad (6.7)$$

where τ_V and ε_V are the stress and strain vectors respectively. The 6×6 matrix E (in 3D, 4×4 in 2D) is the matrix of elastic coefficients. The stress and strain vectors are defined as

$$\tau_V = (\tau_x \ \tau_y \ \tau_z \ \tau_{xy} \ \tau_{yz} \ \tau_{xz})^T \quad (6.8)$$

and

$$\varepsilon_V = (\varepsilon_x \ \varepsilon_y \ \varepsilon_z \ 2\varepsilon_{xy} \ 2\varepsilon_{yz} \ 2\varepsilon_{xz})^T. \quad (6.9)$$

In 2D the stress vector is

$$\tau_V = (\tau_x \ \tau_y \ \tau_z \ \tau_{xy})^T \quad (6.10)$$

and the strain vector

$$\varepsilon_V = (\varepsilon_x \ \varepsilon_y \ \varepsilon_z \ 2\varepsilon_{xy})^T. \quad (6.11)$$

When plane stress computation is requested $\tau_z = 0$, otherwise $\varepsilon_z = 0$. Cylindrically symmetric case is identical to the 2D case, the components are given in the order of r , z , and ϕ . The matrix E is given as input for the anisotropic material model of Elmer.

6.2.2 Modal, harmonic and stability analysis

In addition to steady state and time dependent equations, modal, harmonic and stability analysis may be considered. In modal analysis the Fourier transform of the homogeneous form of the dynamical equation is

$$\rho\omega^2\vec{\phi} = \nabla \cdot \tau(\vec{\phi}), \quad (6.12)$$

or

$$\omega^2 \int_{\Omega} \rho \phi_k \psi_k \, d\Omega = \int_{\Omega} \tau_{ij}(\vec{\phi}) \epsilon_{ij}(\vec{\psi}) \, d\Omega, \quad (6.13)$$

where ω is the angular frequency and $\vec{\phi}$ is the corresponding vibration mode.

When modal analysis of pre-stressed solids are considered, we first perform a steady analysis to compute stress tensor, here denoted by σ_{ij} , and solve the variational equation

$$\omega^2 \int_{\Omega} \rho \phi_k \psi_k \, d\Omega = \int_{\Omega} \tau_{ij}(\vec{\phi}) \epsilon_{ij}(\vec{\psi}) \, d\Omega + \int_{\Omega} \sigma_{ij} \frac{\partial \phi_k}{\partial x_i} \frac{\partial \psi_k}{\partial x_j} \, d\Omega. \quad (6.14)$$

The last term on the right-hand-side represents here the geometric stiffness due to external loads, thermal stresses etc.

In stability analysis the buckling modes $\vec{\phi}$ are obtained from

$$-\lambda \int_{\Omega} \sigma_{ij} \frac{\partial \phi_k}{\partial x_i} \frac{\partial \psi_k}{\partial x_j} \, d\Omega = \int_{\Omega} \tau_{ij}(\vec{\phi}) \epsilon_{ij}(\vec{\psi}) \, d\Omega, \quad (6.15)$$

where λ is the margin of safety with respect to bifurcation (the current load can be multiplied by factor λ before stability is lost).

The equations may be interpreted as generalized eigenproblems and solved with standard techniques.

6.2.3 Rayleigh damping

Damping may be taken into consideration using viscous damping or Rayleigh damping, in which it is assumed that the damping matrix C is proportional to the mass M and stiffness matrices K , or

$$C = \alpha M + \beta K \quad (6.16)$$

The identification of suitable damping coefficients α and β may be a difficult task.

6.2.4 Boundary conditions

For each boundary either a Dirichlet boundary condition

$$d_i = d_i^b \quad (6.17)$$

or a force boundary condition

$$\tau \cdot \vec{n} = \vec{g} \quad (6.18)$$

must be given. The default boundary condition is the natural boundary condition which implies that $\vec{g} = 0$.

The user may give spring k or damping λ coefficients on the boundary. These enable the introduction of the force term in the form

$$\vec{g} = k\vec{d} + \lambda \frac{\partial \vec{d}}{\partial t} \quad (6.19)$$

which may be solved implicitly maintaining the linear form of the equation.

6.2.5 Model lumping

For linear structures it is possible to create a lumped model that gives the same dependence between force and displacement as the original distributed model,

$$F = KD \quad (6.20)$$

where $F = (F_x F_y F_z M_x M_y M_z)^T$ and $D = (D_x D_y D_z \phi_x \phi_y \phi_z)^T$. However, the lumped model is not uniquely defined as it depends on the force or displacement distribution used in the model lumping. In the current model lumping procedure the lumping is done with respect to a given boundary. The lumped force and momentum are then integrals over this boundary,

$$F_i = \int_A f_i dA. \quad (6.21)$$

Lumped displacements and angles are determined as the mean values over the boundary,

$$D_i = \frac{1}{A} \int_A d_i dA. \quad (6.22)$$

Therefore the methodology works best if the boundary is quite rigid in itself.

There are two different model lumping algorithms. The first one uses pure lumped forces and lumped moments to define the corresponding displacements and angles. In 3D this means six different permutations. Each permutation gives one row of the inverse matrix K^{-1} . Pure lumped forces are obtained by constant force distributions whereas pure moments are obtained by linearly varying loads vanishing at the center of area. Pure moments are easily achieved only for relatively simple boundaries which may limit the usability of the model lumping utility.

The second choice for model lumping is to set pure translations and rotations on the boundary and compute the resulting forces on the boundary. This method is not limited by geometric constraints. Also here six permutations are required to get the required data. In this method the resulting matrix equation is often better behaving as in the model lumping by pure forces which may be a reason another reason to favour this procedure.

6.3 Keywords

`Solver solver id`

Note that all the keywords related to linear solver (starting with `Linear System`) may be used in this solver as well. They are defined elsewhere.

`Equation String [StressSolver]`

A describing name for the solver. This can be changed but it must be given,

`Procedure File "StressSolve" "StressSolver"`

Name of the solver subroutine.

`Eigen Analysis Logical`

Modal or stability analysis may be requested with this keyword.

`Eigen System Values Integer`

The number of the lowest eigen states must be given with this keyword, if modal or stability analysis is in effect.

`Harmonic Analysis Logical`

Time-harmonic analysis where the solution becomes complex if damping is defined. The solution algorithm assumes that the diagonal entries in the matrix equation dominates.

`Frequency Real`

The frequency related to the harmonic analysis. If the simulation type is `scanning` this may a scalar function, otherwise it is assumed to be a vector of the desired frequencies.

`Displace Mesh Logical`

Should the mesh be deformed by the displacement field. The default is `True` except for eigen and harmonic analysis.

`Stability Analysis Logical`

If set to `true`, then eigen analysis is stability analysis. Otherwise modal analysis is performed.

`Geometric Stiffness Logical`

If set to `true`, then geometric stiffness is taken into account in modal analysis.

`Calculate Strains Logical`

Computes the strain tensor of the solution.

`Calculate Stresses Logical`

If set to `true` the stress tensor will be computed. Also von Mises will be computed by default.

`Calculate Principal Logical`

Computes the principal stress components.

`Calculate Pangle Logical`

Calculate the principal stress angles.

`Model Lumping Logical`

If model lumping is desired this flag should be set to `True`.

`Model Lumping Filename File`

The results from model lumping are saved into an external file the name of which is given by this keyword.

`Fix Displacements Logical`

This keyword defined if the displacements or forces are set and thereby chooses the model lumping algorithm.

`Constant Bulk System Logical`

For some type of analysis only the boundary conditions change from one subroutine call to another. Then the original matrix may be maintained using this logical keyword. The purpose is mainly to save time spent on matrix assembly.

Update Transient System Logical

Even if the matrix is defined constant it may change with time. The time may also be pseudo-time and then for example the frequency could change with time thus making the harmonic system different between each timestep. This keyword has effect only if the previous keyword is also defined to be true.

Equation eq id

The equation section is used to define a set of equations for a body or set of bodies:

Stress Analysis Logical

if set to True, solve the Navier equations.

Plane Stress Logical

If set to True, compute the solution according to the plane stress situation $\tau_{zz} = 0$. Applies only in 2D.

Body Force bf id

The body force section may be used to give additional force terms for the equations.

Stress Bodyforce 1 Real

Stress Bodyforce 2 Real

Stress Bodyforce 3 Real

The keywords may be used to give volume force.

Stress Bodyforce 1 im Real

Stress Bodyforce 2 im Real

Stress Bodyforce 3 im Real

The keywords may be used to give volume force for the imaginary part. May be applied only to harmonic solution of the equation.

Stress Load Real

Keyword for defining stress load for the body.

Strain Load Real

Keyword for defining strain load for the body.

Initial Condition ic id

The initial condition section may be used to set initial values for the field variables. The following variables are active:

Displacement i Real

For each displacement component $i = 1, 2, 3$.

Material mat id

The material section is used to give the material parameter values. The following material parameters may be set in Navier equations.

Density Real The value of density is given with this keyword. The value may be constant, or variable.

Poisson Ratio Real

For isotropic materials Poisson ratio must be given with this keyword.

Youngs Modulus Real

The elastic modulus must be given with this keyword. The modulus may be given as a scalar for the isotropic case or as 6×6 (3D) or 4×4 (2D and axisymmetric) matrix for the anisotropic case. Although the matrices are symmetric, all entries must be given.

Rayleigh Damping Logical

Apply rayleig damping.

Rayleigh Damping Alpha Real

Rayleigh Damping Beta Real

The parameters of Rayleigh damping.

Pre Stress Real

One may give prestress as an input to the solver.

Pre Strain Real

One may give prestrain as an input to the solver.

Heat Expansion Coefficient Real

If thermal stresses are to be computed this keyword may be used to give the value of the heat expansion coefficient. May also be given as 3×3 tensor for 3D cases, and 2×2 tensor for 2D cases.

Reference Temperature Real

If thermal stresses are to be computed this keyword may be used to give the value of the reference temperature of the stress free state.

Rotate Elasticity Tensor Logical

For anisotropic materials the principal directions of anisotropy do not always correspond to the coordinate axes. Setting this keyword to True enables the user to input Youngs Modulus matrix with respect to the principal directions of anisotropy. Otherwise Youngs Modulus should be given with respect to the coordinate axis directions.

Material Coordinates Unit Vector 1(3) Real [1 0 0]

Material Coordinates Unit Vector 2(3) Real [0 0.7071 0.7071]

Material Coordinates Unit Vector 3(3) Real [0 -0.7071 0.7071]

The above vectors define the principal directions of the anisotropic material. These are needed only if Rotate Elasticity Tensor is set to True. The values given above define the direction of anisotropy to differ from the coordinate axes by a rotation of 45 degrees about x-axis, for example.

Mesh Velocity 1 Real

Mesh Velocity 2 Real

Mesh Velocity 3 Real

Keywords for giving the mesh velocity

Boundary Condition bc id

The boundary condition section holds the parameter values for various boundary condition types. Dirichlet boundary conditions may be set for all the primary field variables. The one related to Navier equations are

Displacement i Real

Dirichlet boundary condition for each displacement component $i = 1, 2, 3$.

Normal-Tangential Displacement Logical

The Dirichlet conditions for the vector variables may be given in normal-tangential coordinate system instead of the coordinate axis directed system. The first component will in this case be the normal component and the components 2,3 two orthogonal tangent directions.

Normal Force Real

A force normal to the boundary is given with this keyword.

Force i Real

A force in the given in coordinate directions $i = 1, 2, 3$.

Force i Im Real

An imaginary part of the force in the given in coordinate directions $i = 1, 2, 3$. Applies only to harmonic simulation.

Normal Force Im Real

A imaginary part of the force normal to the boundary is given with this keyword. Applies only to harmonic simulation.

Damping Real

Introduces a force proportional to velocity with the given coefficient. Also Damping i and Damping ij may be given.

Spring Real

Introduces a force proportional to displacement with the given coefficient. Also Spring i and Spring ij may be given.

Stress Load Real

Keyword for defining stress load for the boundary.

Model Lumping Boundary Logical True

When using the model lumping utility the user must define which boundary is to be loaded in order to determined the lumped model.

Model 7

Finite Elasticity

Module name: ElasticSolve

Module subroutines: ElasticSolver

Module authors: Mikko Lyly, Juha Ruokolainen, Mika Malinen

Document authors: Mika Malinen

Document edited: March 11, 2018

7.1 Introduction

This chapter is concerned with the equations which describe finite deformations of elastic solids. As the region of space occupied by the body at time t is not known in advance, it is not convenient to handle the equations in the form that expresses the field equations on the deformed configuration. Therefore the associated problem is formulated here by employing the reference configuration which equals to the region occupied by the body before the deformation.

7.2 Field equations

Let Ω denote the reference configuration, so that the region of space occupied by the body at the time t is given by

$$\Omega_t = \mathbf{x}(\Omega, t),$$

with $\mathbf{x}(\cdot, t)$, for fixed t , a deformation of Ω . If we define the displacement $\mathbf{u}(\mathbf{p}, t)$ of the material point $\mathbf{p} \in \Omega$ by

$$\mathbf{u}(\mathbf{p}, t) = \mathbf{x}(\mathbf{p}, t) - \mathbf{p},$$

the basic system of field equations describing finite deformations of the body Ω may then be written as

$$\begin{aligned}\rho_0 \ddot{\mathbf{u}} - \text{Div } \mathbf{S} &= \mathbf{b}_0, \\ \mathbf{S} &= \mathbf{F} \bar{\Sigma}(\mathbf{C}), \\ \mathbf{F} &= \mathbf{I} + \nabla \mathbf{u}, \quad \mathbf{C} = \mathbf{F}^T \mathbf{F},\end{aligned}\tag{7.1}$$

where ρ_0 gives the density when the body is in the reference position, the tensor field \mathbf{S} is referred to as the first Piola-Kirchhoff stress, and $\mathbf{b}_0 = \mathbf{b}_0(\mathbf{p}, t)$ defines a body force. The response function $\bar{\Sigma}(\mathbf{C})$ generally characterizes the second Piola-Kirchhoff stress as a function of the right Cauchy-Green tensor \mathbf{C} .

In the basic case it is assumed that either

$$\bar{\Sigma}(\mathbf{C}) = \frac{\lambda}{2} [\text{tr}(\mathbf{C} - \mathbf{I})] \mathbf{I} + \mu(\mathbf{C} - \mathbf{I})\tag{7.2}$$

or, when the neo-Hookean material is assumed,

$$\bar{\Sigma}(\mathbf{C}) = \frac{\lambda}{2}[\det \mathbf{C} - 1]\mathbf{C}^{-1} + \mu(\mathbf{I} - \mathbf{C}^{-1}), \quad (7.3)$$

with λ and μ the Lamé material parameters. We note that a common way to express (7.2) uses the material strain tensor

$$\mathbf{E} = 1/2(\mathbf{C} - \mathbf{I}), \quad (7.4)$$

so that the constitutive law (7.2) may be written as

$$\Sigma = \lambda \text{tr}(\mathbf{E})\mathbf{I} + 2\mu\mathbf{E}.$$

To treat the incompressible neo-Hookean material associated with the limit case $\lambda \rightarrow \infty$ (or equivalently $\nu \rightarrow 1/2$, with ν the Poisson ratio), we introduce an auxiliary field p defined by

$$(1/\lambda)p = -\frac{1}{2}[\det \mathbf{C} - 1] \quad (7.5)$$

to replace (7.3) by

$$\bar{\Sigma}(\mathbf{C}, p) = -p\mathbf{C}^{-1} + \mu(\mathbf{I} - \mathbf{C}^{-1}). \quad (7.6)$$

To handle the case of a nearly incompressible material (the value of ν close to 0.5) computationally, the field p is taken to be an additional unknown which is solved under the constraint (7.5). It is notable that in the limit case of incompressible material p is unique only up to a constant if the displacement is prescribed over the entire boundary of the body (a special care is then needed to ensure that the condition $\det \mathbf{C} = \det \mathbf{F}^2 = 1$ is respected by the boundary conditions).

The solver also offers some utilities to handle more general constitutive laws so that a user-defined material model in the form of an Abaqus user subroutine (UMAT) may be included. In this case the constitutive law has to be defined differently by employing the Cauchy stress σ whose material description $\sigma_m(\mathbf{p}, t) \equiv \sigma(\mathbf{x}(\mathbf{p}, t), t)$ is related to the first Piola-Kirchhoff stress by

$$\mathbf{S} = (\det \mathbf{F})\sigma_m \mathbf{F}^{-T}. \quad (7.7)$$

A user-defined subroutine should define a stress response function $\bar{\sigma}$ to specify how the Cauchy stress depends on a strain field $\hat{\mathbf{E}}(\mathbf{F})$ and an additional N -tuple of state variable fields denoted by $\mathbf{q} = (q_1, \dots, q_N)$. One may then consider a generic constitutive law of the type

$$\sigma_m(\mathbf{p}, t) = \bar{\sigma}(\hat{\mathbf{E}}(\mathbf{F})(\mathbf{p}, t), \mathbf{q}(\mathbf{p}, t)). \quad (7.8)$$

The user-defined subroutine should also define the derivative of the Cauchy stress at the current state with respect to the strain variable for performing the nonlinear solution procedure. That is, in the differentiation the stress response function is treated as the composition

$$\mathbf{F} \mapsto \bar{\sigma}(\cdot, \mathbf{q}) \circ \hat{\mathbf{E}}(\mathbf{F}), \quad (7.9)$$

with \mathbf{q} being taken as a parameter, so that the Fréchet differential of (7.9) is obtained by using the chain rule as

$$\mathbf{U} \mapsto D\bar{\sigma}(\hat{\mathbf{E}}(\mathbf{F}), \mathbf{q})[D\hat{\mathbf{E}}(\mathbf{F})[\mathbf{U}]]. \quad (7.10)$$

The user-defined subroutine must thus return a representation of $D\bar{\sigma}(\hat{\mathbf{E}}(\mathbf{F}), \mathbf{q})$ so that the differential (7.10) can be generated on the side where the user-defined subroutine is called. In this connection the strain field $\hat{\mathbf{E}}(\mathbf{F})$ need not necessarily be the material strain (7.4), as the Hencky (logarithmic) strain or the standard linearized strain can also be used. Including a user-defined material model requires some programming skills and the best place to check for the current functionality is the solver code. A template subroutine for a user-defined material model can also be found in the file `UMATLib.F90` that is located in the same directory as the code of the solver module.

7.3 Boundary conditions

Boundary conditions may be obtained by prescribing the displacement and surface traction on complementary parts Γ_1 and Γ_2 of the boundary $\partial\Omega$, respectively. The displacement boundary condition is simply of the form

$$\mathbf{u} = \hat{\mathbf{u}}(\mathbf{p}, t), \quad (7.11)$$

with $\hat{\mathbf{u}}$ a prescribed vector field on $\Gamma_1 \times [0, T]$.

Handling surface traction is more involved. First, assume that the surface traction vector \mathbf{s} on the deformed surface $\mathbf{x}(\Gamma_2, t)$ is normal to the tangent plane of the deformed boundary surface, so that

$$\mathbf{s}(\mathbf{x}, t) = g(\mathbf{x}, t)\mathbf{m}(\mathbf{x}),$$

where $\mathbf{m}(\mathbf{x})$ is the unit normal on the deformed configuration, $\mathbf{x} \in \mathbf{x}(\Gamma_2, t)$ for any t , and $g(\mathbf{x}, t)$ is a given scalar function. This can be shown to be equivalent to specifying the values of $\mathbf{S}\mathbf{n}$ such that

$$\mathbf{S}\mathbf{n} = \hat{g}(\det \mathbf{F})\mathbf{F}^{-T}\mathbf{n} \quad \text{on } \Gamma_2 \times [0, T], \quad (7.12)$$

where $\mathbf{n} = \mathbf{n}(\mathbf{p})$ is the normal vector to the boundary $\partial\Omega$ and $\hat{g} = \hat{g}(\mathbf{p}, t) = g(\mathbf{x}(\mathbf{p}, t), t)$. The constraint (7.12) gives rise to a nonlinear force term which is handled in the computational solution iteratively by using a lagged-value approximation.

The surface traction \mathbf{s} may also be specified by giving its components with respect to the frame of reference such that

$$\mathbf{s}(\mathbf{x}(\mathbf{p}, t), t) = \hat{\mathbf{s}}(\mathbf{p}, t), \quad (7.13)$$

with $\hat{\mathbf{s}}(\mathbf{p}, t)$ a given vector. While the condition (7.13) specifies the actual force per unit area of the deformed surface, it also possible to specify directly the pseudo-traction $\mathbf{s}^0 = \mathbf{S}\mathbf{n}$ which gives the actual force per unit undeformed area. If the pseudo-traction is specified on Γ_2 as

$$\mathbf{s}^0(\mathbf{p}, t) = \hat{\mathbf{s}}^0(\mathbf{p}, t), \quad (7.14)$$

the total force exerted across Γ_2 is then given by the surface integral

$$\int_{\Gamma_2} \hat{\mathbf{s}}^0 d\Gamma.$$

If the alternate (7.13) is used, the total force is obtained by

$$\int_{\Gamma_2} \hat{\mathbf{s}}(\det \mathbf{F}) \sqrt{\mathbf{n} \cdot (\mathbf{F}^{-1} \mathbf{F}^{-T}) \mathbf{n}} d\Gamma$$

where the additional scalar term in the integrand relates to the area change during the deformation.

7.4 Linearization: The basic constitutive laws

To handle the model computationally, the constitutive law $\mathbf{S} = \hat{\mathbf{S}}(\mathbf{F}) = \mathbf{F}\boldsymbol{\Sigma}(\mathbf{F})$, with $\boldsymbol{\Sigma}(\mathbf{F}) = \bar{\boldsymbol{\Sigma}}(\mathbf{F}^T \mathbf{F})$, has to be linearized also. This can be done in terms of the derivative $D\hat{\mathbf{S}}(\mathbf{F})[\mathbf{U}]$ by using the Newton approximation

$$\hat{\mathbf{S}}(\mathbf{F}_{k+1}) \approx \hat{\mathbf{S}}(\mathbf{F}_k) + D\hat{\mathbf{S}}(\mathbf{F}_k)[\mathbf{F}_{k+1} - \mathbf{F}_k].$$

We then have

$$\hat{\mathbf{S}}(\mathbf{F}_{k+1}) \approx \hat{\mathbf{S}}(\mathbf{F}_k) + \mathbf{F}_k D\boldsymbol{\Sigma}(\mathbf{F}_k)[\mathbf{F}_{k+1} - \mathbf{F}_k] + (\mathbf{F}_{k+1} - \mathbf{F}_k)\boldsymbol{\Sigma}(\mathbf{F}_k).$$

In view of $\mathbf{F}_{k+1} - \mathbf{F}_k = \nabla \mathbf{u}_{k+1} - \nabla \mathbf{u}_k$, this leads to the linearization

$$\begin{aligned} \hat{\mathbf{S}}(\mathbf{F}_{k+1}) &\approx \hat{\mathbf{S}}(\mathbf{F}_k) + \mathbf{F}_k D\boldsymbol{\Sigma}(\mathbf{F}_k)[\nabla \mathbf{u}_{k+1} - \nabla \mathbf{u}_k] + (\nabla \mathbf{u}_{k+1} - \nabla \mathbf{u}_k)\boldsymbol{\Sigma}(\mathbf{F}_k) \\ &= \hat{\mathbf{S}}(\mathbf{F}_k) - \mathbf{F}_k D\boldsymbol{\Sigma}(\mathbf{F}_k)[\nabla \mathbf{u}_k] - \nabla \mathbf{u}_k \boldsymbol{\Sigma}(\mathbf{F}_k) + \mathbf{F}_k D\boldsymbol{\Sigma}(\mathbf{F}_k)[\nabla \mathbf{u}_{k+1}] + \nabla \mathbf{u}_{k+1} \boldsymbol{\Sigma}(\mathbf{F}_k). \end{aligned} \quad (7.15)$$

In the case of (7.2) the derivative of the response function Σ is given by

$$D\Sigma(\mathbf{F})[\nabla \mathbf{v}] = \frac{\lambda}{2} \text{tr}(\mathbf{F}^T \nabla \mathbf{v} + \nabla \mathbf{v}^T \mathbf{F}) \mathbf{I} + \mu(\mathbf{F}^T \nabla \mathbf{v} + \nabla \mathbf{v}^T \mathbf{F}),$$

while

$$D\Sigma(\mathbf{F})[\nabla \mathbf{v}] = \lambda(\det \mathbf{F})^2 \text{tr}(\nabla \mathbf{v} \mathbf{F}^{-1}) \mathbf{C}(\mathbf{F})^{-1} + \\ [\mu - \frac{\lambda}{2}(\det \mathbf{F} - 1)(\det \mathbf{F} + 1)] \mathbf{C}(\mathbf{F})^{-1} [\mathbf{F}^T \nabla \mathbf{v} + \nabla \mathbf{v}^T \mathbf{F}] \mathbf{C}(\mathbf{F})^{-1}$$

for the neo-Hookean material obeying (7.3). In the computation of the associated tangential stiffness matrix, which result from substituting the approximation (7.15) into the discrete version of the weak formulation of (7.1), the following self-adjointness property

$$\mathbf{F}_k D\Sigma(\mathbf{F}_k)[\nabla \mathbf{u}_{k+1}] \cdot \nabla \mathbf{v} + \nabla \mathbf{u}_{k+1} \Sigma(\mathbf{F}_k) \cdot \nabla \mathbf{v} = \mathbf{F}_k D\Sigma(\mathbf{F}_k)[\nabla \mathbf{v}] \cdot \nabla \mathbf{u}_{k+1} + \nabla \mathbf{v} \Sigma(\mathbf{F}_k) \cdot \nabla \mathbf{u}_{k+1}$$

is also used.

When a nearly incompressible neo-Hookean material is considered, we need to linearize both the constitutive law $\mathbf{S} = \hat{\mathbf{S}}(\mathbf{F}, p) = \mathbf{F} \Sigma(\mathbf{F}, p)$, with $\Sigma(\mathbf{F}, p) = \bar{\Sigma}(\mathbf{F}^T \mathbf{F}, p)$, and the constraint

$$\varphi(\mathbf{F}, p) = 0 \quad (7.16)$$

where

$$\varphi(\mathbf{F}, p) = \varepsilon p + 1/2(\det \mathbf{F})^2 - 1/2 \quad (7.17)$$

with

$$\varepsilon = 1/\lambda. \quad (7.18)$$

The Newton updates related to solving (7.16) are given by

$$(\det \mathbf{F}_k)^2 \text{tr}[(\mathbf{F}_{k+1} - \mathbf{F}_k) \mathbf{F}_k^{-1}] + \varepsilon(p_{k+1} - p_k) = -\varphi(\mathbf{F}_k, p_k). \quad (7.19)$$

To obtain the Newton linearization of the stress response function, we note in particular that the derivative of the function $\mathbf{G}(\mathbf{F}, p) = -p\mathbf{C}(\mathbf{F})^{-1}$ is given by

$$D\mathbf{G}(\mathbf{F}, p)[(\mathbf{U}, h)] = -h\mathbf{C}(\mathbf{F})^{-1} + p\mathbf{C}(\mathbf{F})^{-1}[\mathbf{F}^T \mathbf{U} + \mathbf{U}^T \mathbf{F}] \mathbf{C}(\mathbf{F})^{-1}.$$

It then follows that

$$D\Sigma(\mathbf{F}, p)[(\mathbf{U}, h)] = -h\mathbf{C}(\mathbf{F})^{-1} + (p + \mu)\mathbf{C}(\mathbf{F})^{-1}[\mathbf{F}^T \mathbf{U} + \mathbf{U}^T \mathbf{F}] \mathbf{C}(\mathbf{F})^{-1}. \quad (7.20)$$

7.5 Linearization: The case of a user-defined material model

Other constitutive laws can be defined by including a user-defined material model in the form of an Abaqus user subroutine (UMAT). In contrast to the other constitutive laws, such subroutine is expected to return the Cauchy stress corresponding to the current estimate of the strain increment with respect to the previous converged solution (the solution before the time/load increment) as well as the derivative of the stress response function for computing the differential (7.10).

It should be noted that, despite relying on the Cauchy stress, the weak formulation is still generated from the equilibrium equations expressed in terms of the first Piola-Kirchhoff stress. The stress power calculations are thus done in terms of an energetically conjugate pair of the first Piola-Kirchhoff stress and the rate of change of the deformation gradient. To this end, in view of (7.7)–(7.9), we consider the mapping

$$\mathbf{F} \mapsto \hat{\mathbf{S}}(\mathbf{F}) = (\det \mathbf{F}) \{ \bar{\sigma}(\cdot, \mathbf{q}) \circ \hat{\mathbf{E}}(\mathbf{F}) \} \mathbf{F}^{-T} \quad (7.21)$$

and use (7.10) to obtain the Fréchet differential

$$D\hat{\mathbf{S}}(\mathbf{F})[\mathbf{U}] = (\det \mathbf{F}) \{ D\bar{\sigma}(\hat{\mathbf{E}}(\mathbf{F}), \mathbf{q}) [D\hat{\mathbf{E}}(\mathbf{F})[\mathbf{U}]] \} \mathbf{F}^{-T} \\ + (\det \mathbf{F}) \text{tr}(\mathbf{U} \mathbf{F}^{-1}) \bar{\sigma}(\hat{\mathbf{E}}(\mathbf{F}), \mathbf{q}) \mathbf{F}^{-T} - (\det \mathbf{F}) \bar{\sigma}(\hat{\mathbf{E}}(\mathbf{F}), \mathbf{q}) \mathbf{F}^{-T} \mathbf{U}^T \mathbf{F}^{-T}. \quad (7.22)$$

In the differentiation the state variables are thus treated as parameters. In the case of a user-defined material model the equation (7.22) provides the key ingredient for linearization.

7.6 Stress and strain computation

In addition to solving for the displacement, the solver can produce the strain and stress fields associated with the solution. In this connection the strain tensor is defined by (7.4). In the stress computation the material description of the usual Cauchy stress σ is produced. That is, we measure the surface force per unit area in the deformed configuration and write $\sigma_m(\mathbf{p}, t) = \sigma(\mathbf{x}(\mathbf{p}, t), t)$. We note that this stress is related to one of the Piola-Kirchhoff stresses as

$$\sigma_m = (\det \mathbf{F})^{-1} \mathbf{S} \mathbf{F}^T = (\det \mathbf{F})^{-1} \mathbf{F} \bar{\Sigma}(\mathbf{C}) \mathbf{F}^T. \quad (7.23)$$

7.7 Keywords

Simulation

In specifying the keywords for the simulation section, note that all coordinate systems are not supported.

Coordinate System String

The coordinate system may be Cartesian 2D, Cartesian 3D or Axi Symmetric.

Material mat id

The following keywords relate to giving the material parameters for the finite elasticity solver.

Density Real

This keyword is used for defining the density field ρ_0 corresponding to the reference configuration.

Poisson Ratio Real

The values of the scalar Lamé material parameters depend on the Poisson ratio as in the case of the linear elasticity solver. The Poisson ratio is given by using this keyword.

Youngs Modulus Real

The values of the scalar Lamé material parameters depend on the Youngs modulus as in the case of the linear elasticity solver. This keyword specifies the value of the Youngs modulus.

UMAT Subroutine File

The value of this keyword consists of two string parameters. The first parameter specifies the name of a compiled file containing the definition of a user-defined material model in the UMAT form. The second parameter defines the name of the user-defined subroutine. If this keyword is used, all material models must be defined in the UMAT form.

Number of Material Constants Integer

The value of this keyword defines the number of material constants that are passed to the UMAT subroutine.

Material Constants Real

The values of the material constants passed to the UMAT subroutine.

Number of State Variables Integer

This keyword is used to declare the number of state variables.

Name String

This keyword may be used to define the name argument of the UMAT subroutine.

Equation eq id

Plane Stress Logical

If the coordinate system is chosen to be Cartesian 2D, this keyword may be used to activate nonlinear plane stress analysis. In the case of plane stress the definition of the Lamé parameter λ is altered such that the plane stress components are directly obtained in terms of the plane strain components. The strain E_{33} can then be expressed as $E_{33} = -\nu/(1 - \nu)(E_{11} + E_{22})$.

Solver solver id

Equation String [ElasticSolver]

A describing name for the solver. This can be changed but it must be given,

Procedure File "ElasticSolve" "ElasticSolver"

Name of the solver subroutine.

Neo-Hookean Material Logical

By default the constitutive law (7.2) is employed. Switching to the neo-Hookean material model (7.3) can be performed by giving the value `True` for this keyword.

Mixed Formulation Logical

This keyword is used to handle incompressible or nearly incompressible material obeying the neo-Hookean constitutive law. If the value `True` is given for this keyword, the field p is taken to be an additional unknown which is solved under the constraint (7.5). In this case the solver assumes that the mesh files correspond to the lowest-order finite elements (the lowest-order pressure approximation together with the second-order displacement approximation is then constructed by default). In addition, the default names for the displacement variable \mathbf{u} and pressure variable p are then `Disp` and `Pres`, respectively.

Calculate Strains Logical

If the value `True` is given for this keyword, the strains are also computed. The strain components are output into an ordered six-tuple as $(E_{xx} E_{yy} E_{zz} E_{xy} E_{yz} E_{xz})$. However, in the axially symmetric simulation only four components are produced as $(E_{xx} E_{zz} E_{yy} E_{xy})$, with the convention $x = r$ and $z = \theta$.

Calculate Stresses Logical

If the value `True` is given for this keyword, the Cauchy stress (7.23) is also computed. The stress components are output into an ordered six-tuple in the same way as the strain.

Calculate Principal Logical

If the strain or stress computation is activated, this keyword can be used to activate the computation of principal components.

Calculate PAngle Logical

This keyword can be used to activate the computation of the principal angles for the stress tensor. If the value `True` is given for this keyword, then the computation of principal components is also activated.

Initialize State Variables Logical

If the material model is defined in terms of a user-defined material model (UMAT), an extra call of the UMAT subroutine can be done to obtain the state variables in the initial state (stress-free initial condition is supposed).

Body Force bf id

This section may be used to define body forces.

Inertial Bodyforce j Real

This keyword may be used to give the component j of the body force $\mathbf{b}(\mathbf{x}, t)$ in order to define $\mathbf{b}_0 = \rho_0(\mathbf{p})\mathbf{b}(\mathbf{x}(\mathbf{p}, t), t)$. It is noted that in this case $\mathbf{b}(\mathbf{x}, t)$ defines the body force per unit mass. The density changes are then considered correctly, i.e. the condition $\rho(\mathbf{x}(\mathbf{p}, t), t) \det \mathbf{F}(\mathbf{p}, t) = \rho_0(\mathbf{p})$ is respected.

Stress Bodyforce j Real

This keyword may be used to give the component j of the body force $\mathbf{b}(\mathbf{x}, t)$ in order to define $\mathbf{b}_0 = (\det \mathbf{F})\mathbf{b}(\mathbf{x}(\mathbf{p}, t), t)$. It is noted that $\mathbf{b}(\mathbf{x}, t)$ is now the body force per unit volume of the deformed body, so this type of force is appropriate for specifying true volumetric forces, whatever they might be.

Boundary Condition `bc id`

The Dirichlet conditions (7.11) for the displacement variable of the solver can be given in the standard manner. Other options for defining boundary conditions are explained in the following.

Normal Surface Traction `Real`

A surface force which is normal to the deformed boundary and gives force per unit area of the deformed surface may be given with this keyword.

Surface Traction `k Real`

By default this keyword may be used to give the actual force per unit area of the deformed surface. The value of this keyword then specifies the component k of \hat{s} in (7.13). If the keyword command `Pseudo-Traction = True` is also given, then the values of this keyword command are used to determine the components of the pseudo-traction vector \hat{s}^0 which gives the actual force per unit undeformed area.

Pseudo-Traction `Logical`

If this keyword has the value `True`, then the surface force is defined via the pseudo-traction condition; see the explanation of the keyword `Surface Traction k` below.

FSI BC `Logical`

If this keyword has the value `True`, then the Navier–Stokes flow solution is used to determine the surface force generated by the flow.

Model 8

Shell Equations of Classical Elasticity

Module name: ShellSolver

Module subroutines: ShellSolver

Module authors: Mika Malinen

Document authors: Mika Malinen

Document edited: Jul 13, 2018

8.1 Introduction

This chapter is concerned with the equations which describe deformations of thin elastic shells. Here a shell refers to a curved three-dimensional body which can be described in terms of its mid-surface and thickness (the extent of the shell in the direction of normal to the mid-surface). When the placement of the shell in its reference configuration is described by using a system of normal coordinates (that is, the third coordinate curve is perpendicular to two coordinate curves on the mid-surface), simplifications to solving 3-D elasticity equations can be sought via the process of dimensional reduction, so that unknowns that depend only on the two curvilinear coordinates associated with the shell mid-surface can be employed. Classical shell theory [3] is dedicated to the study of such models by assuming that the exact parametrization of the shell mid-surface is known in advance.

However, classical shell theory cannot often be applied in a straightforward manner in connection with finite element modelling, since in practice the mapping giving the mid-surface is not usually available in an explicit form. To offer generality, the shell solver described here creates a computational surface model by assuming that information about the surface position and the director vector (the unit normal to the exact mid-surface) are given at the nodes of a background mesh. Elementwise approximations of the mid-surface are then created such that the surface position and the normal to the approximate surface agree with the data given at the nodes. It should be noted that the approximate mid-surface obtained in this way generally gives a more accurate description of the surface position than what would be obtained by using the standard strategy where straightforward Lagrange interpolation merely based on the nodal position data is employed. Here the approximation of the director field is then derived via straightforward differentiation of the mapping giving the approximate surface (it should be noted that this approximation is consistent with the given data at the nodes).

In the first place each (physical) element S of the reconstructed mid-surface is originally parametrized in terms of the rectangular Cartesian coordinates of points of a usual reference element (a square or an equilateral triangle). However, the solver considered performs additional computation in order to find a convenient elementwise reparametrization in terms of lines of curvature coordinates, so that we may write $S = \varphi_K(K)$ with $\varphi_K : K \subset \mathbb{R}^2 \rightarrow \mathbb{E}^3$ and the rectangular Cartesian coordinates y^α of a point \mathbf{y} of K corresponding to lines of curvature coordinates on S (for the basic concepts related to lines of curvature, see [2]). Then each point on the surface can naturally be associated with an orthogonal system of basis vectors which offer a convenient starting point for representing vector-valued fields over the surface. Since the basis is orthogonal, the components of a vector field have intuitive physical significance and tensor

calculations related to the shell equations are greatly simplified in comparison with the case of general curvilinear coordinates.

8.2 Discrete shell model

The shell model we employ is related to the simplest kinematic assumption that enables the approximation of 3-D elasticity equations without additional assumptions about the state of stress. The kinematic assumption conforms with using solid finite elements which have nodes located on the upper and lower surfaces of the shell, together with auxiliary degrees of freedom to enable a normal strain field that depends linearly on the normal coordinate. Conventional 2-D shell models can then be derived by imposing the condition of vanishing normal stress, but use of a refined shell model can also be considered in this setting.

8.2.1 Preliminaries

We may now think of a physical surface element $S = \varphi_K(K)$ to be associated with the physical solid element $\Omega_S \subset \mathbf{E}^3$ which is the image of the set $S \times [-d/2, d/2]$ under a mapping of normal coordinates

$$(\mathbf{p}, y^3) \mapsto \mathbf{p} + y^3(\mathbf{a}_3 \circ \varphi_K^{-1})(\mathbf{p}), \quad (8.1)$$

with d being the shell thickness and the surface function $\mathbf{p} \mapsto (\mathbf{a}_3 \circ \varphi_K^{-1})(\mathbf{p})$ giving the unit normal to the mid-surface at a point $\mathbf{p} \in S$ in terms of the vector field $\mathbf{a}_3 : K \rightarrow \mathbb{R}^3$. Since the normal coordinates $(\mathbf{p}, y^3) \in S \times [-d/2, d/2]$ thus identify a point in the physical element Ω_S , it is natural to approximate the elementwise restriction of the displacement vector field of the shell

$$\mathbf{u} : S \times [-d/2, d/2] \rightarrow \mathbb{R}^3, \quad (\mathbf{p}, y^3) \mapsto \mathbf{u}(\mathbf{p}, y^3), \quad (8.2)$$

in a systematic manner such that

$$\mathbf{u}(\varphi_K(\mathbf{y}), y^3) \equiv \hat{\mathbf{u}}(\mathbf{y}, y^3) \equiv \mathbf{v}^{(0)}(\mathbf{y}) - y^3 \mathbf{v}^{(1)}(\mathbf{y}) - \frac{1}{2}(y^3)^2 \mathbf{v}^{(2)}(\mathbf{y}), \quad (8.3)$$

with the vector fields $\mathbf{v}^{(k)} : K \rightarrow \mathbb{R}^3$ in two variables being taken as unknowns. The mathematical domain of definition for three-dimensional shell variables will thus be the set $\Omega_K = K \times [-d/2, d/2]$ whose points are mapped to the points of the physical space \mathbf{E}^3 as

$$(\mathbf{y}, y^3) \mapsto \Theta(\mathbf{y}, y^3) \equiv \varphi_K(\mathbf{y}) + y^3 \mathbf{a}_3(\mathbf{y}). \quad (8.4)$$

This representation of geometry follows by writing an alternate referential description of the normal coordinates representation defined by (8.1).

Each physical point $\Theta(\mathbf{y}, y^3)$ of the shell can be associated with three vectors

$$\mathbf{g}_k(\mathbf{y}, y^3) = D\Theta(\mathbf{y}, y^3)[\hat{\mathbf{e}}_k] = \partial_k \Theta(\mathbf{y}, y^3), \quad (8.5)$$

with $\hat{\mathbf{e}}_k$ being the orthonormal basis vectors associated with the mathematical domain of definition, to give a covariant basis for the translation space \mathbb{R}^3 of \mathbf{E}^3 . Similarly, by restricting to the mid-surface, we define a set of surface basis vectors $\mathbf{a}_i : K \rightarrow \mathbb{R}^3$, which give the covariant basis at $\mathbf{p} = \varphi_K(\mathbf{y})$, via

$$\mathbf{a}_\alpha(\mathbf{y}) = D\varphi_K(\mathbf{y})[\hat{\mathbf{e}}_\alpha] = \partial_\alpha \varphi_K(\mathbf{y}), \quad \mathbf{a}_3(\mathbf{y}) \cdot \mathbf{a}_\alpha(\mathbf{y}) = 0. \quad (8.6)$$

The covariant components of the metric surface tensor \mathbf{A} (the first fundamental form) are now given by

$$A_{\alpha\beta}(\mathbf{y}) = \mathbf{a}_\alpha(\mathbf{y}) \cdot \mathbf{a}_\beta(\mathbf{y}). \quad (8.7)$$

We also define $B_{\alpha\beta} : K \rightarrow \mathbb{R}$ by

$$B_{\alpha\beta}(\mathbf{y}) = \mathbf{a}_3(\mathbf{y}) \cdot \partial_\alpha \mathbf{a}_\beta(\mathbf{y}) = -\mathbf{a}_\alpha(\mathbf{y}) \cdot \partial_\beta \mathbf{a}_3(\mathbf{y}). \quad (8.8)$$

They give the covariant components of the second fundamental form B of the surface at $\mathbf{p} = \varphi_K(\mathbf{y})$. We shall also need the Christoffel symbols which are defined by

$$\Gamma_{\alpha\beta}^\gamma(\mathbf{y}) = \partial_\beta \mathbf{a}_\alpha(\mathbf{y}) \cdot \mathbf{a}^\gamma(\mathbf{y}) \quad (8.9)$$

where the contravariant basis vectors \mathbf{a}^i of the surface satisfy the orthogonality conditions

$$\mathbf{a}_i(\mathbf{y}) \cdot \mathbf{a}^j(\mathbf{y}) = \delta_i^j, \quad (8.10)$$

with δ_i^j being the Kronecker's symbol.

When lines of curvature coordinates are used, the two sets of basis vectors are related by

$$\mathbf{g}_1(\mathbf{y}, y^3) = \mu_1^1(\mathbf{y}, y^3) \mathbf{a}_1(\mathbf{y}), \quad \mathbf{g}_2(\mathbf{y}, y^3) = \mu_2^2(\mathbf{y}, y^3) \mathbf{a}_2(\mathbf{y}), \quad \mathbf{g}_3(\mathbf{y}, y^3) = \mathbf{a}_3(\mathbf{y}) \quad (8.11)$$

where μ_α^β are the mixed components of a diagonal (shifter) tensor. We then have

$$\mu_1^1(\mathbf{y}, y^3) = 1 + y^3/R_1(\mathbf{y}), \quad \mu_2^2(\mathbf{y}, y^3) = 1 + y^3/R_2(\mathbf{y}) \quad (8.12)$$

with

$$R_1(\mathbf{y}) = -\frac{B_{11}(\mathbf{y})}{A_{11}(\mathbf{y})} = -B_1^1(\mathbf{y}) \quad \text{and} \quad R_2(\mathbf{y}) = -\frac{B_{22}(\mathbf{y})}{A_{22}(\mathbf{y})} = -B_2^2(\mathbf{y}) \quad (8.13)$$

being the principal radii of curvature. The sign convention is here chosen such that $R_\alpha > 0$ if the normal vector is directed away from the centre of curvature.

The simplest consistent kinematic assumption corresponds to the choice where the nontrivial part of the second-order contribution $\mathbf{v}^{(2)}$ can be expressed simply as

$$\mathbf{v}^{(2)}(\mathbf{y}) = [\mathbf{v}^{(2)}(\mathbf{y}) \cdot \mathbf{a}_3(\mathbf{y})] \mathbf{a}^3(\mathbf{y}), \quad (8.14)$$

i.e. only one scalar field describes the second-order part with respect to y^3 . Writing all the unknowns in terms of the contravariant basis vectors of the surface yields (cf. the assumption (8.3))

$$\hat{\mathbf{u}}(\mathbf{y}, y^3) = \sum_{i=1}^3 [v_i(\mathbf{y}) - y^3 \beta_i(\mathbf{y}) - \frac{1}{2} (y^3)^2 \psi_i(\mathbf{y})] \mathbf{a}^i(\mathbf{y}), \quad (8.15)$$

where $\psi_1(\mathbf{y}) = \psi_2(\mathbf{y}) = 0$. Our kinematic assumption thus involves seven scalar fields which are functions of the mid-surface coordinates (for original contributions that use this assumption, see historical notes in [3] and papers by Reissner and Naghdi mentioned there). The assumption that only one quadratic component in y^3 is included is motivated by having the ability to expand all components of linearized strain tensor up to the first-order terms in y^3 .

8.2.2 The measure of strain

The measure of three-dimensional strain we employ is based on the Green-St Venant strain tensor field $\mathbf{E}(\hat{\mathbf{u}}) : \Omega_K \rightarrow \text{Sym}$ associated with the elementwise restriction $\hat{\mathbf{u}}$ of the displacement field. Its components are defined such that

$$2\bar{E}_{ij}(\hat{\mathbf{u}})(\cdot) = \mathbf{g}_i(\cdot) \cdot \partial_j \hat{\mathbf{u}}(\cdot) + \partial_i \hat{\mathbf{u}}(\cdot) \cdot \mathbf{g}_j(\cdot) + [\partial_i \hat{\mathbf{u}}(\cdot)] \cdot [\partial_j \hat{\mathbf{u}}(\cdot)] \quad (8.16)$$

and hence measure the change of the metric tensor associated with the displacement field. In practice, we perform a change of basis in order to express the strain tensor field as

$$\mathbf{E}(\hat{\mathbf{u}})(\mathbf{y}, y^3) = E_{ij}(\hat{\mathbf{u}})(\mathbf{y}, y^3) \mathbf{a}^i(\mathbf{y}) \otimes \mathbf{a}^j(\mathbf{y}),$$

so that the components are then expressed with respect to the surface basis vectors depending on the two curvilinear coordinates of the mid-surface only. Alternatively, the most basic representation of strain follows by switching to a local orthonormal basis $\{\mathbf{e}_1(\mathbf{y}), \mathbf{e}_2(\mathbf{y}), \mathbf{e}_3(\mathbf{y})\} = \{\mathbf{e}^1(\mathbf{y}), \mathbf{e}^2(\mathbf{y}), \mathbf{e}^3(\mathbf{y})\}$ obtained as

$$\mathbf{e}_\alpha(\mathbf{y}) = \frac{\mathbf{a}_\alpha(\mathbf{y})}{A_\alpha(\mathbf{y})}, \quad \mathbf{e}_3(\mathbf{y}) = \mathbf{a}_3(\mathbf{y}), \quad \text{with } A_\alpha(\mathbf{y}) = |\mathbf{a}_\alpha(\mathbf{y})|, \quad (8.17)$$

and writing then

$$\mathbf{E}(\hat{\mathbf{u}})(\mathbf{y}, y^3) = \hat{E}_{ij}(\hat{\mathbf{u}})(\mathbf{y}, y^3) \mathbf{e}^i(\mathbf{y}) \otimes \mathbf{e}^j(\mathbf{y}).$$

The components of these representations can be shown to obey the following transformation rules (here the summation convention is adopted so that a repeated index simultaneously appearing as a subscript and as a superscript in a term imply summation over all possible values, with a Greek index being however allowed to have values in $\{1, 2\}$)

$$\begin{aligned} \hat{E}_{\alpha\beta}(\hat{\mathbf{u}}) &= \frac{E_{\alpha\beta}(\hat{\mathbf{u}})}{A_\alpha A_\beta} = \frac{(\mu^{-1})_\alpha^\lambda (\mu^{-1})_\beta^\nu \bar{E}_{\lambda\nu}(\hat{\mathbf{u}})}{A_\alpha A_\beta}, \quad \hat{E}_{\alpha 3}(\hat{\mathbf{u}}) = \frac{E_{\alpha 3}(\hat{\mathbf{u}})}{A_\alpha} = \frac{(\mu^{-1})_\alpha^\nu \bar{E}_{\nu 3}(\hat{\mathbf{u}})}{A_\alpha}, \\ \hat{E}_{33}(\hat{\mathbf{u}}) &= E_{33}(\hat{\mathbf{u}}) = \bar{E}_{33}(\hat{\mathbf{u}}). \end{aligned}$$

By letting $\mathbf{U} = (v_1, v_2, \dots)$ to denote an n -tuple of 2-D scalar fields which determine $\hat{\mathbf{u}}$, we next expand the components of the strain tensor in power series in terms of the normal coordinate y^3 to write

$$E_{ij}(\hat{\mathbf{u}}(\mathbf{U}))(\mathbf{y}, y^3) = \gamma_{ij}(\mathbf{U})(\mathbf{y}) + \epsilon_{ij}(\mathbf{U})(\mathbf{y}) - y^3 \kappa_{ij}(\mathbf{U})(\mathbf{y}) - y^3 \chi_{ij}(\mathbf{U})(\mathbf{y}) \quad (8.18)$$

where the 2-D fields $\gamma_{ij}(\mathbf{U}) : K \rightarrow \mathbb{R}$ and $\kappa_{ij}(\mathbf{U}) : K \rightarrow \mathbb{R}$ are linear with respect to \mathbf{U} , while $\epsilon_{ij}(\mathbf{U}) : K \rightarrow \mathbb{R}$ and $\chi_{ij}(\mathbf{U}) : K \rightarrow \mathbb{R}$ are nonlinear. If the shell undergoes only small deflections, linearization can additionally be performed by omitting the nonlinear terms to write then

$$E_{ij}(\hat{\mathbf{u}}(\mathbf{U})) = \gamma_{ij}(\mathbf{U}) - y^3 \kappa_{ij}(\mathbf{U}).$$

We note that the tangent plane components $\gamma_{\alpha\beta}(\mathbf{U})$ and $\kappa_{\alpha\beta}(\mathbf{U})$ constitute the so-called membrane strain and bending strain tensors, while $\gamma_{\alpha 3}(\mathbf{U})$ are the transverse shear strains.

By using the notion of the covariant derivatives (see, for example, [2])

$$v_{\beta|\alpha} \equiv \partial_\alpha v_\beta - \Gamma_{\alpha\beta}^\nu v_\nu, \quad v_{3|\alpha} \equiv \partial_\alpha v_3 \quad (8.19)$$

to have

$$\partial_\alpha(v_i \mathbf{a}^i) = (v_{\beta|\alpha} - B_{\alpha\beta} v_3) \mathbf{a}^\beta + (v_{3|\alpha} + B_\alpha^\beta v_\beta) \mathbf{a}^3,$$

the components of the linearized membrane strain tensor are found to be

$$\gamma_{\alpha\beta}(\mathbf{U}) = \frac{1}{2}(v_{\alpha|\beta} + v_{\beta|\alpha}) - B_{\alpha\beta} v_3, \quad (8.20)$$

while the components of the linearized bending strain tensor and the transverse shear strain tensor are

$$\begin{aligned} \kappa_{11}(\mathbf{U}) &= \beta_{1|1} - B_1^1 A_{11} \beta_3 - B_1^1 \gamma_{11}(\mathbf{U}), \\ \kappa_{22}(\mathbf{U}) &= \beta_{2|2} - B_2^2 A_{22} \beta_3 - B_2^2 \gamma_{22}(\mathbf{U}), \\ \kappa_{12}(\mathbf{U}) &= \frac{1}{2}(\beta_{1|2} + \beta_{2|1} - B_1^1 v_{2|1} - B_1^1 v_{1|2}) \end{aligned} \quad (8.21)$$

and

$$\begin{aligned} 2\gamma_{13}(\mathbf{U}) &= \partial_1 v_3 + B_1^1 v_1 - \beta_1, \\ 2\gamma_{23}(\mathbf{U}) &= \partial_2 v_3 + B_2^2 v_2 - \beta_2. \end{aligned} \quad (8.22)$$

The parts of the linearized transverse shear strains that depend linearly on y^3 are given by

$$\begin{aligned} 2\kappa_{13}(\mathbf{U}) &= \partial_1 \beta_3 + \psi_1 - 2B_1^1 \gamma_{13}(\mathbf{U}), \\ 2\kappa_{23}(\mathbf{U}) &= \partial_2 \beta_3 + \psi_2 - 2B_2^2 \gamma_{23}(\mathbf{U}). \end{aligned} \quad (8.23)$$

It should be noted that under the kinematic assumption the expression for the linearized normal strain is precisely

$$\gamma_{33}(\mathbf{U}) - y^3 \kappa_{33}(\mathbf{U}) = -\beta_3 - y^3 \psi_3 \quad (8.24)$$

so that we cannot generally go beyond linear terms in y^3 in the expansions of strains. This generally motivates our choice to truncate the expressions for strains as done in (8.18). In addition, the terms of type $B_\beta^\alpha \gamma_{\alpha k}(\mathbf{U})$ in the expressions for $\kappa_{ij}(\mathbf{U})$ are expected to have little impact on the strain energy of a thin shell and are therefore omitted to obtain simplified expressions.

The nonlinear fields $\epsilon_{ij}(\mathbf{U})$ can be expressed as

$$\begin{aligned} 2\epsilon_{11}(\mathbf{U}) &= (\gamma_{11}(\mathbf{U}))^2 A^{11} + (v_{2|1})^2 A^{22} + (v_{3|1} + B_1^1 v_1)^2, \\ 2\epsilon_{22}(\mathbf{U}) &= (v_{1|2})^2 A^{11} + (\gamma_{22}(\mathbf{U}))^2 A^{22} + (v_{3|2} + B_2^2 v_2)^2, \\ 2\epsilon_{12}(\mathbf{U}) &= \gamma_{11}(\mathbf{U}) v_{1|2} A^{11} + \gamma_{22}(\mathbf{U}) v_{2|1} A^{22} + (v_{3|1} + B_1^1 v_1)(v_{3|2} + B_2^2 v_2), \\ 2\epsilon_{13}(\mathbf{U}) &= -\gamma_{11}(\mathbf{U}) \beta_1 A^{11} - v_{2|1} \beta_2 A^{22} - (v_{3|1} + B_1^1 v_1) \beta_3, \\ 2\epsilon_{23}(\mathbf{U}) &= -v_{1|2} \beta_1 A^{11} - \gamma_{22}(\mathbf{U}) \beta_2 A^{22} - (v_{3|2} + B_2^2 v_2) \beta_3, \\ 2\epsilon_{33}(\mathbf{U}) &= \beta_1^2 A^{11} + \beta_2^2 A^{22} + \beta_3^2. \end{aligned} \quad (8.25)$$

In the current version of the shell solver all nonlinear components $\chi_{ij}(\mathbf{U})$ are omitted. This is motivated by an observation that the omission of ψ_α implies the expressions of $\chi_{ij}(\mathbf{U})$ being incomplete, i.e. a later addition of higher-order terms ψ_α would alter $\chi_{ij}(\mathbf{U})$. The lack of completeness is similarly true in regard to the expressions (8.23).

It is notable that the derivatives of ψ_i do not occur in the expressions for strains when only the terms up to the first order with respect to y^3 are taken into account. When weak solutions of shell equations are sought, the unknowns ψ_i are thus seen to be exceptional in that less regularity (a weak solution in L_2) can be supposed. In addition, since the only quadratic component ψ_3 appears only in the expression for $\kappa_{33}(\mathbf{U})$, a variational formulation of the shell problem may not necessitate solving ψ_3 as tightly coupled with the other unknowns, i.e. its values can be found afterwards when $\mathbf{v}^{(k)}$, $k = 0, 1$, have first been solved.

8.2.3 The principle of virtual work

To simplify the statement of the principle of virtual work, we now shorten the expressions for the strain components by omitting the splitting into the linear and nonlinear parts, so that

$$\mathbf{E}(\hat{\mathbf{u}}(\mathbf{U})) = \boldsymbol{\varepsilon}(\mathbf{U}) - y^3 \boldsymbol{\rho}(\mathbf{U}) \quad (8.26)$$

with

$$\begin{aligned} \varepsilon_{ij}(\mathbf{U}) &\equiv \gamma_{ij}(\mathbf{U}) + \epsilon_{ij}(\mathbf{U}), \\ \rho_{ij}(\mathbf{U}) &\equiv \kappa_{ij}(\mathbf{U}) + \chi_{ij}(\mathbf{U}). \end{aligned} \quad (8.27)$$

Currently the shell solver can handle only a nonlinear extension of the standard constitutive law for an isotropic material characterized by Young's modulus E and Poisson's ratio ν . That is, we assume that

$$\hat{\Sigma}^{ij}(\cdot) = \frac{\nu E}{(1+\nu)(1-2\nu)} [\hat{E}_{11}(\hat{\mathbf{u}})(\cdot) + \hat{E}_{22}(\hat{\mathbf{u}})(\cdot) + \hat{E}_{33}(\hat{\mathbf{u}})(\cdot)] \delta_{ij} + \frac{E}{1+\nu} \hat{E}_{ij}(\hat{\mathbf{u}})(\cdot) \quad (8.28)$$

where the scalar fields $\hat{\Sigma}^{ij} : \Omega_K \rightarrow \mathbb{R}$ are the components of the second Piola-Kirchhoff stress with respect to the orthonormal basis. This model is expected to be feasible when the stretches of the shell remain relatively small, while rigid body deformations of arbitrary magnitude are possible.

The statement of the three-dimensional principle of virtual work can now be put into the form

$$DU(\mathbf{U})[\mathbf{V}] = \mathcal{Q}(\mathbf{U}, \mathbf{V})$$

where $DU(\mathbf{U})[\mathbf{V}]$ gives the derivative of strain energy and $\mathcal{Q}(\mathbf{U}, \cdot)$ is a linear functional determined by loads. We note that the linear functional does not always depend on the solution \mathbf{U} . In that case we could simplify the statement of the principle of virtual work as

$$DU(\mathbf{U})[\mathbf{V}] = \mathcal{L}(\mathbf{V}),$$

with $\mathcal{L}(\cdot)$ being a linear functional independent of \mathbf{U} . The strain energy is expressed elementwise as

$$\mathcal{U}_K(\mathbf{U}) = \int_{\Omega_K} W(\mathbf{E}(\hat{\mathbf{u}}(\mathbf{U}))(\mathbf{y}, y^3)) \sqrt{g(\mathbf{y}, y^3)} d\mathbf{y} dy^3, \quad (8.29)$$

with $W : \text{Sym} \rightarrow \mathbb{R}$ giving the strain-energy density and $g(\mathbf{y}, y^3)$ denoting the determinant of three-dimensional metric tensor. The contribution to the principle of virtual work can then be expressed as

$$\int_{\Omega_K} \mathcal{R}(\mathbf{E}(\hat{\mathbf{u}}(\mathbf{U}))(\mathbf{y}, y^3)) \cdot D\mathbf{E}(\hat{\mathbf{u}}(\mathbf{U}))[\mathbf{V}](\mathbf{y}, y^3) \sqrt{g(\mathbf{y}, y^3)} d\Omega_K = \mathcal{Q}_K(\mathbf{U}, \mathbf{V}) \quad (8.30)$$

for all kinematically admissible \mathbf{V} , with the second Piola-Kirchhoff stress field $\boldsymbol{\Sigma}(\cdot) = \mathcal{R}(\mathbf{E}(\hat{\mathbf{u}}(\mathbf{U}))(\cdot))$ being found as the derivative $\mathcal{R}(\mathbf{E}) \cdot \mathbf{H} = DW(\mathbf{E})[\mathbf{H}]$.

A truly two-dimensional variational formulation follows by using (8.26) in connection with (8.29) and performing the integration over the thickness of the shell. In the integration, we have chosen to neglect terms of $O(d/R_\alpha)$ in order to simplify the final statement of the 2-D version of the principle of virtual work. After some reduction, the strain energy over K may be expressed as

$$\begin{aligned} \mathcal{U}_K(\mathbf{U}) = & \frac{Ed}{2(1-\nu^2)} \int_K \left\{ \nu[\varepsilon_{11}(\mathbf{U}) + \varepsilon_{22}(\mathbf{U})]^2 + (1-\nu) \sum_{\alpha,\beta=1}^2 [\varepsilon_{\alpha\beta}(\mathbf{U})]^2 \right\} \sqrt{a} dK \\ & + \frac{Ed(1-\nu)}{2(1+\nu)(1-2\nu)} \int_K \left\{ \varepsilon_{33}(\mathbf{U}) + \frac{\nu}{1-\nu} [\varepsilon_{11}(\mathbf{U}) + \varepsilon_{22}(\mathbf{U})] \right\}^2 \sqrt{a} dK \\ & + \frac{Ed}{1+\nu} \int_K [(\varepsilon_{13}(\mathbf{U}))^2 + (\varepsilon_{23}(\mathbf{U}))^2] \sqrt{a} dK \\ & + \frac{Ed^3}{24(1-\nu^2)} \int_K \left\{ \nu[\rho_{11}(\mathbf{U}) + \rho_{22}(\mathbf{U})]^2 + (1-\nu) \sum_{\alpha,\beta=1}^2 [\rho_{\alpha\beta}(\mathbf{U})]^2 \right\} \sqrt{a} dK \\ & + \frac{Ed^3}{12(1+\nu)} \int_K [(\rho_{13}(\mathbf{U}))^2 + (\rho_{23}(\mathbf{U}))^2] \sqrt{a} dK \end{aligned} \quad (8.31)$$

with a being the determinant of the metric surface tensor.

8.2.4 Compatibility relations

While finite element approximation of 2-D shell equations is usually considered in a circumstance where the entire shell is described by using a single chart $\varphi : \bar{\omega} \rightarrow \mathbb{E}^3$, with $\bar{\omega} \subset \mathbb{R}^2$ partitioned into finite elements, here we however have a situation where each element is associated with an independent chart. Thus, in order to make this approach practical, we need to define appropriate compatibility relations between local finite element approximations, which are constructed to be functions of the elementwise coordinates y^α of the sets K . To this end, it is convenient to assume a set of global degrees of freedom which correspond to the components of a spatial displacement field and its directional derivatives in the normal direction with respect to the constant basis vectors associated with the global coordinate frame. The set of global degrees of freedom is thus selected such that they determine the nodal values of $\mathbf{v}^{(k)}$ as being the following vectors and directional derivatives associated with the spatial displacement field

$$\begin{aligned} \mathbf{v}^{(0)}(\mathbf{y}_k) &= \mathbf{v}(\mathbf{p}_k, 0), \\ \mathbf{v}^{(1)}(\mathbf{y}_k) &= -D\mathbf{v}(\mathbf{p}_k, 0)[\mathbf{d}_k], \\ \mathbf{v}^{(2)}(\mathbf{y}_k) &= -D^2\mathbf{v}(\mathbf{p}_k, 0)[\mathbf{d}_k, \mathbf{d}_k], \end{aligned} \quad (8.32)$$

where \mathbf{y}_k denotes a node on K , with $\mathbf{p}_k = \varphi_K(\mathbf{y}_k)$ the corresponding point in the physical space, and $\mathbf{d}_k = \mathbf{a}_3(\mathbf{y}_k)$ gives the unit normal to the mid-surface at point \mathbf{p}_k .

8.2.5 Strain reduction operators

Efficient and reliable finite element discretization of shell equations is a challenging task and unsettled questions still remain. Shell problems can exhibit different types of asymptotic behaviour when the shell thickness tends to zero [1]. A great challenge for a finite element designer is to work out a formulation which works for all possible asymptotic scenarios. An ultimate challenge would be to accompany the method with a mathematical error analysis covering the full versatility of shell problems.

For the above reasons applying standard finite elements is not an option unless basis functions of high degree are used, as standard low-order methods are not suitable for approximating fields with negligible membrane and transverse shear strains (such case corresponds a bending-dominated asymptotic behaviour). This relates to a computational trouble known as finite element locking. To obtain better low-order methods, the shell solver of Elmer employs strain reduction operators which are applied to the membrane and transverse shear strains and designed, in the first place, to relax constraints that arise in the case of bending-dominated problems.

Currently the strain reduction operators have been worked out for the lowest-order approximation only (a 3-node triangle or 4-node quadrilateral).

8.3 Specifying surface data

In order to approximate the shell mid-surface, the shell solver needs information about the director vector. As alternatives the nodal director data can be read from a file, or it can be associated with an ordinary Elmer variable "Director" that has been solved before executing the shell solver. It should be noted that parallel versions of file formats used in connection with reading from files do not exist yet, so at the moment parallel computation is possible only when the director field is made available as the Elmer variable "Director".

The nodal director data can be formatted into special data files in two ways. The first option is to write a file `mesh.director` which lists the director at the nodes in a similar way as nodes are defined in the file `mesh.nodes`. That is, the contents of the file `mesh.director` should be organized as

```
n1 dx dy dz
n2 dx dy dz
...
nn dx dy dz
```

The first integer is the identification number for the node followed by three real numbers which are the components of the director with respect to the global coordinate frame. The file should be located in the same place as the standard mesh files.

The second option is to provide a file `mesh.elements.data` which should define the nodal director in elementwise manner and associate the name 'director' with this data. Thus, if just the director data is given, the contents of the file should be arranged as

```
element: element_id_1
director: dx_1 dy_1 dz_1 ... dx_n dy_n dz_n
end
element: ...
...
end
```

Here the nodewise ordering of the director data on lines starting with `director:` must correspond to that of the `mesh.elements` file. Also this file should be located in the same place as the standard mesh files. It is noted that a file `mesh.elements.data` is considered first in priority. Optionally, given a file `mesh.director`, the solver can write the director data as elementwise property to a file whose format conforms with a file `mesh.elements.data`. The elementwise data contained in `mesh.elements.data` may generally be discontinuous over adjacent finite elements.

One way to create the Elmer variable "Director" is to utilize the solver `NormalSolver` that uses the background mesh for the computation of the normal vector. This approach can compromise the accuracy of the geometry model but allows parallel computation. For special cases where the dependency of the

director on the global coordinates is known, a slight modification of the `NormalSolver` module might be enough for obtaining both an accurate approximation of the director and having the option to run the shell solver in parallel.

8.4 Keywords

Simulation

Coordinate System String Cartesian 3D

The coordinate system should be selected to be three-dimensional, although basis functions for computation correspond to 2-D finite elements.

Material mat id

The following keywords relate to specifying the shell thickness and material parameters.

Shell Thickness Real

The thickness d of the shell is specified with this keyword.

Poisson Ratio Real

Poisson's ratio is given by using this keyword.

Youngs Modulus Real

This keyword specifies the value of Young's modulus.

Density Real

This keyword is used for defining the density of the material.

Solver solver id

Equation String

A describing name for the solver.

Procedure File "ShellSolver" "ShellSolver"

The name of the solver subroutine.

Variable String Deflection[U:3 DNU:3]

The name of the solver variable can be chosen freely (but it must be used consistently elsewhere).

Variable DOFs Integer 6

There is no need for using this keyword as the only possible value is 6 and it is given automatically by the solver. The first three component of the solver variable define the mid-surface displacement field $\boldsymbol{v}^{(0)}$ (the default variable name U), while the rest are related to the vector $\boldsymbol{v}^{(1)}$ (the default variable name DNU). In this connection the components of both the vectors are defined with respect to the global coordinate frame.

Large Deflection Logical

By default the nonlinear equations are solved. With the value being `False`, the linearized strain tensor is employed.

Displace Mesh Logical

If this keyword is given the value `True`, the mesh is mapped to represent the deformed configuration. The formulation of the total Lagrangian type is nevertheless used, so the shell problem is posed over the undeformed configuration.

Nonlinear System Convergence Tolerance Real

The ratio of the 2-norm of the nonlinear system residual to the 2-norm of the initial right-hand side vector is always used as the stopping criterion for the nonlinear iteration. This keyword specifies the stopping tolerance for the Newton iteration to solve the nonlinear system.

Linear System Convergence Tolerance *Real*

It should be noted that each linearized problem solved during the nonlinear iteration gives an increment $\delta \mathbf{U}^{k+1} = \mathbf{U}^{k+1} - \mathbf{U}^k$ to the previous iterate. Therefore, except for the case of solving the linear shell model (`Large Deflection = False`), a rather mild stopping tolerance may often be used for the linear systems without affecting the progress of the nonlinear iteration.

Strain Reduction Operator *Integer*

This keyword specifies the choice of strain reduction operators. If this keyword is not given, the solver switches to a method which has been found to give the best results for benchmark cases considered during the development.

Body Force *bf id***Normal Pressure** *Real*

The value of this keyword should give the sum of normal tractions applied to the upper and lower faces of the shell at $y^3 = \pm d$. If the shell model is nonlinear, this is the normal surface force per unit area of the deformed surface. In this case the normal to the deformed mid-surface and the area are computed by using the previous iterate.

Boundary Condition *bc id*

The Dirichlet conditions for the components of $\mathbf{v}^{(0)}$ and $\mathbf{v}^{(1)}$ can be given in the standard manner. Note that here the components of both vectors are defined with respect to the global coordinate frame.

U i *Real*

If the default variable name is used, then, with $i=1, 2, 3$, Dirichlet BCs for the components of the mid-surface displacement $\mathbf{v}^{(0)}$ can be given.

DNU i *Real*

If the default variable name is used, then, with $i=1, 2, 3$, Dirichlet BCs for the components of the mid-surface displacement $\mathbf{v}^{(1)}$ can be given.

Bibliography

- [1] D. Chapelle and K.J. Bathe. *The Finite Element Analysis of Shells - Fundamentals*. Springer, second edition, 2011.
- [2] P.G. Ciarlet. *An Introduction to Differential Geometry with Applications to Elasticity*. Springer, Dordrecht, the Netherlands, 2005.
- [3] P.M. Naghdi. Foundations of elastic shell theory. In *Progress in Solid Mechanics, Vol. 4 (I.N. Sneddon, R. Hill, Eds) North-Holland*, pages 1–90, 1963.

Model 9

Plate Equations of Linear Elasticity

Module name: Smitc

Module subroutines: SmitcSolver

Module authors: Mikko Lyly, Jani Paavilainen

Document authors: Mikko Lyly, Peter Råback

Document created: August 26th 2002

9.1 Introduction

The linear elastic plate elements of Elmer are based on the shear deformable model of Reissner and Mindlin. The finite element discretization is performed using the so called stabilized MITC-plate elements, which are free from numerical locking.

9.1.1 Reissner-Mindlin model

The displacement $\vec{u} = (u_x, u_y, u_z)$ of a Reissner-Mindlin plate (thin or moderately thick linearly elastic body which in its undeformed reference configuration occupies the three dimensional region $\Omega \times (-\frac{t}{2}, \frac{t}{2})$, where Ω is the midsurface and t the thickness) is obtained from the kinematic equations

$$u_x(x, y, z) = -\theta_x(x, y) \cdot z \quad (9.1)$$

$$u_y(x, y, z) = -\theta_y(x, y) \cdot z \quad (9.2)$$

$$u_z(x, y, z) = w(x, y) \quad (9.3)$$

where θ_x and θ_y are components of the rotation vector $\underline{\theta} = (\theta_x, \theta_y)$ and w is the transverse deflection of the mid-surface, see Figure 1.

The functions w and $\underline{\theta} = (\theta_x, \theta_y)$ are determined from the condition that they minimize the total potential energy

$$\frac{1}{2} \int_{\Omega} \underline{\kappa} : \underline{m} \, d\Omega + \int_{\Omega} \underline{\gamma} \cdot \underline{q} \, d\Omega - \int_{\Omega} p w \, d\Omega \quad (9.4)$$

where p is the transverse pressure load, $\underline{\kappa} = \frac{1}{2}(\underline{\nabla}\underline{\theta} + \underline{\nabla}\underline{\theta}^T)$ is the curvature of the mid-surface, $\underline{\gamma} = \underline{\nabla}w - \underline{\theta}$ is the transverse shear strain, $\underline{m} = \underline{\mathcal{E}} : \underline{\kappa}$ is the bending moment, and $\underline{q} = \underline{\mathcal{G}} \cdot \underline{\gamma}$ the transverse shear force vector. The fourth order tensor $\underline{\mathcal{E}}$ and second order tensor $\underline{\mathcal{G}}$ define the bending and shear rigidities of the cross section, respectively. For linearly elastic materials we have $\underline{\mathcal{G}} \cdot \underline{\gamma} = Gt\underline{\gamma}$ and

$$\underline{\mathcal{E}} : \underline{\kappa} = K[\underline{\kappa} + \frac{\nu}{1-\nu}(\text{tr}\underline{\kappa})\underline{I}] \quad (9.5)$$

where $K = Et^3/[12(1-\nu^2)]$ is the bending stiffness, E is Young's modulus, G shear modulus, and ν Poisson ratio. The design of the tensors $\underline{\mathcal{E}}$ and $\underline{\mathcal{G}}$ for orthotropic and perforated materials is discussed in section 9.3.

The minimizer of the energy satisfies the equilibrium equations

$$\underline{\nabla} \cdot \underline{m} + \underline{q} = 0 \quad (9.6)$$

$$-\underline{\nabla} \cdot \underline{q} = p \quad (9.7)$$

9.1.2 Surface tension

When surface tension is present, the following term is added to the energy:

$$\frac{1}{2} \int_{\Omega} \underline{\nabla} w \cdot \mathcal{T} \cdot \underline{\nabla} w \, d\Omega \quad (9.8)$$

where \mathcal{T} is a second order tensor representing the given normal force (usually $\mathcal{T} = T \underline{I}$, where T is constant). The equilibrium equation (9.7) is then rewritten as

$$-\underline{\nabla} \cdot (\underline{q} + \mathcal{T} \cdot \underline{\nabla} w) = p \quad (9.9)$$

9.1.3 Boundary conditions

The following boundary conditions can be applied in the Reissner-Mindlin plate model:

- Soft fixed edge: $w = 0$ and $\underline{\theta} \cdot \underline{n} = 0$
- Hard fixed edge: $w = 0$ and $\underline{\theta} = \underline{0}$
- Soft simply supported edge: $w = 0$
- Hard simply supported edge: $w = 0$ and $\underline{\theta} \cdot \underline{t} = 0$
- Free edge: $\underline{m} \cdot \underline{n} = 0$ and $(\underline{q} + \mathcal{T} \cdot \underline{\nabla} w) \cdot \underline{n} = 0$

The boundary conditions can of course be non-homogeneous as well. For fixed and simply supported edges the prescribed values of w , $\underline{\theta}$, $\underline{\theta} \cdot \underline{n}$, and $\underline{\theta} \cdot \underline{t}$, are taken into account on matrix level after finite element discretization. On the free part of the edge, the non-homogeneous case is treated by adding the following terms in the energy:

$$\int_{\Gamma_{free}} q_n w \, d\Gamma + \int_{\Gamma_{free}} \underline{m}_n \cdot \underline{\theta} \, d\Gamma \quad (9.10)$$

where $q_n = \underline{q} \cdot \underline{n}$ and $\underline{m}_n = \underline{m} \cdot \underline{n}$ are prescribed functions.

9.1.4 Kirchhoff plates

When the thickness of the plate is small ($t \ll \text{diam}(\Omega)$), the Reissner-Mindlin model can be considered as a penalty approximation of the classical plate model of Kirchhoff. The Kirchhoff model is obtained from (9.1)-(9.9) by enforcing the constraint $\underline{\gamma} = \underline{0}$. The governing equations are then reduced to

$$K \Delta \Delta w - T \Delta w = p \quad (9.11)$$

9.1.5 Transient and natural mode analysis

A transient plate model is obtained by adding the inertia term $\rho t \ddot{w}$ on the left hand-side of (9.7), (9.9), and (9.11). Here ρ is the density of the material. The natural vibration frequencies and mode shapes are then obtained by taking $p = 0$ and solving the Fourier transformed equations.

9.2 Finite element implementation

The direct minimization of (9.4) using the standard Galerkin finite element method fails due to the well known numerical locking phenomena (the method is unable to deal with the Kirchhoff constraint $\gamma = \underline{0}$, which becomes valid when t is small). In order to avoid locking, Elmer utilizes the so called SMITC (Stabilization and Mixed Interpolation of Tensorial Components) elements, which are known to be optimally convergent and work well under all conditions [4].

The linear element of the SMITC-family was first introduced by Brezzi, Fortin and Stenberg in [2]. The method is defined by replacing the shear energy term in (9.4) by the following numerical modification:

$$\int_{\Omega} \underline{\gamma}_h \cdot \underline{q}_h \, d\Omega \quad (9.12)$$

where $\underline{\gamma}_h$ is called the reduced shear strain (sometimes also referred to as the assumed or substitute shear) and $\underline{q}_h = (t^2 + \alpha h^2)^{-1} \mathcal{G} \cdot \underline{\gamma}_h$ the reduced shear force. Here h is the mesh size (the diameter of the biggest element) and $\alpha > 0$ is a numerical stabilization parameter (typically $\alpha = 0.15$).

The reduced shear $\underline{\gamma}_h$ is defined elementwise such that

$$\underline{\gamma}_{h|K} = (a_K - b_K y, a_K + c_K x) \quad (9.13)$$

for any element K . The parameters a_K , b_K , and c_K , are determined from the conditions

$$\int_E (\underline{\gamma} - \underline{\gamma}_h) \cdot \underline{t} \, ds = 0 \quad (9.14)$$

for every edge E of K . Here \underline{t} is the counterclockwise tangent to E .

It has been shown [3] that the linear SMITC-element is equivalent to the T3BL (Triangle, 3 nodes, Linked Interpolation) element of Xu, Auricchio and Taylor [8, 1], the anisoparametrically interpolated MIN3 element of Tessler and Hughes [7], and the TRIA3 element of MacNeal [5]. We refer to [3] for a more detailed discussion.

9.3 Elastic parameters for perforated plates

In microelectromechanical systems the plate structures are often perforated in order to reduce the squeezed-film damping effect. This has also an effect on the elasticity equation. If there are so many holes that it is not feasible to treat them individually their effect may be homogenized over the whole structure. In practice this means that the original elastic parameters are replaced by effective parameters that take into account the holes. This method was reported by Pedersen et al. [6] and implemented into the solver by Jani Paavilainen.

In the homogenization effective parameters for an orthotropic plate are defined so that the unperforated model approximates the perforated plate. The basic idea is to set the analytical expressions of the deformation energies of the perforated and unperforated plates equal. This method is inherently limited to simple geometries where analytical expressions may be found. So far, only square holes have been implemented in the solver.

The unit cell of a perforated plate may be assumed to consist of one small square plate with side $b - 2a$, and of four beams of length a as shown in Figure 9.1. Using approximate formulas an analytical formula for the deformation energy of the perforated plate is obtained. This has to be equal to the deformation energy of an unperforated orthotropic membrane. From this condition we get a set of equations from which the effective parameters may be solved.

The elasticity tensor has three independent components, $C_{11} = C_{22}$, $C_{12} = C_{21}$, and C_{44} . The expressions for these are [6],

$$C_{11} = C_{22} = \frac{E}{b^2} \left\{ \frac{b(b-2a)}{1-\nu^2} + \frac{a(b-2a)^2}{b} \right\} \quad (9.15)$$

$$C_{12} = C_{21} = \frac{\nu E(b-2a)}{b(1-\nu^2)} \quad (9.16)$$

$$C_{44} = \frac{E}{4b^2(1+\nu)} \left\{ 2b(b-2a) + \frac{12Ka(b-2a)}{bh^3} \right\}. \quad (9.17)$$

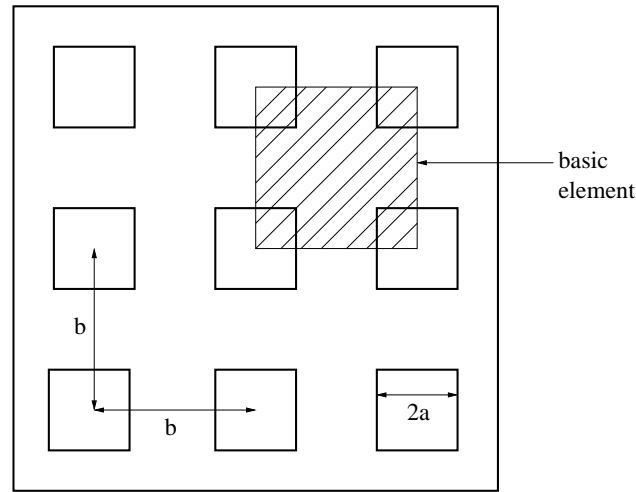


Figure 9.1: The basic element of the perforated plate consisting of five rectangular beams

where K is a constant¹, defined as

$$K = \begin{cases} \frac{1}{3} \left(1 - 0.63 \frac{b-2a}{h} \right) (b-2a)^3 h, & \text{jos } h > b-2a \\ \frac{1}{3} \left(1 - 0.63 \frac{h}{b-2a} \right) (b-2a) h^3, & \text{jos } h < b-2a. \end{cases} \quad (9.18)$$

The midplane tension of the perforated plate may be reduced to lateral stresses of the orthotropic plate by a simple scaling,

$$T = \sqrt{(1 - 4a^2/b^2)} T_0, \quad (9.19)$$

where is the tension T_0 of the perforated plate. Using this reduced tension and the modified material parameters of equations (9.15), (9.16) and (9.17) the orthotropic plate mimics the behavior of the perforated plate when looking at macroscopic quantities. However, the model is not suitable for approximating maximum stresses around the holes, for example.

9.4 Keywords

Solver `solver id`

Equation `String` `SmitcSolver`

Procedure `File` `"Smitc"` `"SmitcSolver"`

The procedure which includes the linear plate model.

Variable `String` `Deflection`

This may be of any name as far as it is used consistently also elsewhere.

Variable `DOFs` `Integer` `3`

Degrees of freedom for the deflection. The first degree is the displacement and the two following ones are its derivatives in the direction of the coordinate axis.

Eigen `Analysis` `Logical`

Also the eigenvalues and eigenmodes of the elasticity equation may be computed. This is done automatically by calling a eigensolver after the original equation has been solved. The default is `False`.

¹In article [6] there is an error in the definition of K . In the article there is an expression $(b-2a)/h^3$, which would make K discontinuous at $h = b-2a$.

Eigen System Values Integer

If the eigenvalues are computed this keyword gives the number of eigenmodes to be computed.
The lowest eigenvalues are always solved for.

Hole Correction Logical

If the plate is perforated the holes may be taken into account by a homogenized model. This is activated with this keyword. The default is `False`.

Material `mat id`

Density Real

Density of the plate.

Poisson ratio Real

Youngs modulus Real

The elastic parameters are given with Young's modulus and Poisson ratio.

Thickness Real

Thickness of the plate.

Tension Real

The plate may be pre-stressed.

Hole Size Real

Hole Fraction Real

If `Hole Correction` is `True` the solver tries to find the size and relative fraction of the holes. If these are present the hole is assumed to be a square hole.

Boundary Condition `bc id`

Deflection `i` Real

Dirichlet BC for the components of the deflection, $i=1,2,3$.

Body Force `bf id`

Pressure Real

Possibility for a body forces. For coupled systems there is a possibility to have up to three forces.
The two others are then marked with `Pressure B` and `Pressure C`.

Spring Real

The local spring which results to a local force when multiplied by the displacement.

Damping Real

The local damping which results to a local force when multiplied by the displacement velocity.
The spring and damping may also be defined as material parameters.

Bibliography

- [1] F. Auricchio and R.L. Taylor. A triangular thick plate finite element with an exact thin limit. *Finite Element in Analysis and Design*, 19:57–68, 1995.
- [2] M. Fortin F. Brezzi and R. Stenberg. Error analysis of mixed-interpolated elements for reissner-mindlin plates. *Mathematical Models and Methods in Applied Sciences*, 1:125–151, 1991.
- [3] M. Lyly. On the connection between some linear triangular reissner-mindlin plate bending elements. *Numerische Mathematik*, 85:77–107, 2000.
- [4] M. Lyly and R. Stenberg. Stabilized mitc plate bending elements. In *Advances in Finite Element Techniques (M. Papadrakakis and B.H.V. Topping, eds.) Civil Comp Press.*, pages 11–16, 1994.
- [5] R.H. MacNeal. Derivation of element stiffness matrices by assumed strain distribution. *Nucl. Engrg. Design*, 70:3–12, 1982.

- [6] M. Pedersen, W. Olthuis, and P. Bergveld. On the mechanical behaviour of thin perforated plates and their application in silicon condenser microphones. *Sensors and Actuators*, A 54:6.
- [7] A. Tessler and T.J.R. Hughes. A three-node mindlin plate element with improved transverse shear. *Comp. Meths. Appl. Mech. Engrg.*, 50:71–101, 1985.
- [8] Z. Xu. A thick-thin triangular plate element. *Int. J. Num. Meths. Eng.*, 33:963–973, 1992.

Model 10

One-dimensional Equations for Elastic Beams

Module name: BeamSolver3D

Module subroutines: TimoshenkoSolver

Module authors: Mika Malinen

Document authors: Mika Malinen

Document edited: May 27, 2019

10.1 Introduction

The solver described in this section can be used to solve structural beam equations. The equilibrium equations are expressed in terms of stress resultants \mathbf{N} and \mathbf{M} that represent forces and moments experienced by the cross section of a beam. If the cross section is considered to be an oriented surface with positive unit normal \mathbf{e}_1 , the stress resultant \mathbf{N} is defined by

$$\mathbf{N} = \int_A \boldsymbol{\sigma} \mathbf{e}_1 \, dA \quad (10.1)$$

where $\boldsymbol{\sigma}$ is the stress tensor and A denotes the cross section of the beam. Generally \mathbf{N} is resolved into components with respect to an orthonormal basis $\{\mathbf{e}_1, \mathbf{e}_2, \mathbf{e}_3\}$ associated with a local frame which has its origin at the intersection of the cross section and the axis of the beam. The coordinates of points with respect to this frame are denoted by (y_1, y_2, y_3) with the y_1 -axis being aligned with the axis of the beam, while the other two axes are assumed to coincide with the principal directions of the cross section. The stress resultant $\mathbf{M} = M_1 \mathbf{e}_1 + M_2 \mathbf{e}_2 + M_3 \mathbf{e}_3$ has as its components the bending moments

$$M_2 = \int_A y_3 (\boldsymbol{\sigma} \mathbf{e}_1) \cdot \mathbf{e}_1 \, dA, \quad M_3 = - \int_A y_2 (\boldsymbol{\sigma} \mathbf{e}_1) \cdot \mathbf{e}_1 \, dA, \quad (10.2)$$

while M_1 is a torsional moment. We note that in general accurate modelling of the torsion is not a simple problem and here a rudimentary approximation will only be employed.

10.2 Governing equations

Let us assume that the axis of the beam is described by using the arc length parametrization $s \in [0, L] \mapsto \mathbf{r}(s) \in \mathbb{E}^3$. The equilibrium equations are then given by

$$\begin{aligned} -\mathbf{N}'(s) &= \mathbf{F}(s), \\ -\mathbf{M}'(s) - \mathbf{t}(s) \times \mathbf{N}(s) &= \mathbf{G}(s), \end{aligned} \quad (10.3)$$

where \mathbf{F} and \mathbf{G} are the densities of applied forces and moments per unit length and $\mathbf{t}(s) \equiv \mathbf{e}_1(s)$ is tangential to the beam axis. It should be noted that \mathbf{F} and \mathbf{G} can be considered to include inertial body forces to obtain equations for transient cases.

In the case of Timoshenko's treatment of shear deformation, suitable generalized measures of strain may be expressed as

$$\begin{aligned}\boldsymbol{\varepsilon}(s) &= \mathbf{u}'(s) - \boldsymbol{\theta}(s) \times \mathbf{t}(s), \\ \boldsymbol{\kappa}(s) &= \boldsymbol{\theta}'(s)\end{aligned}\tag{10.4}$$

where $\mathbf{u} : [0, L] \rightarrow \mathbb{R}^3$ is the displacement of the beam axis and the components of $\boldsymbol{\theta} : [0, L] \rightarrow \mathbb{R}^3$ are the so-called rotations. It is noted that the underlying approximation of the displacement \mathbf{u}_{3D} for a generic point $\mathbf{r}(s) + y_2\mathbf{e}_2(s) + y_3\mathbf{e}_3(s)$ of the cross sections is given componentwise by

$$\begin{aligned}\mathbf{u}_{3D}(s; y_2, y_3) \cdot \mathbf{e}_1(s) &= u_1(s) - y_2\theta_3(s) + y_3\theta_2(s), \\ \mathbf{u}_{3D}(s; y_2, y_3) \cdot \mathbf{e}_2(s) &= u_2(s) - y_3\theta_1(s), \\ \mathbf{u}_{3D}(s; y_2, y_3) \cdot \mathbf{e}_3(s) &= u_3(s) + y_2\theta_1(s).\end{aligned}\tag{10.5}$$

The constitutive relations are written as

$$\begin{aligned}\mathbf{N} &= \mathbf{D}\boldsymbol{\varepsilon}, \\ \mathbf{M} &= \mathbf{E}\boldsymbol{\kappa},\end{aligned}\tag{10.6}$$

where

$$\mathbf{D} = \text{diag}(EA, GAk_2, GAk_3)\tag{10.7}$$

and

$$\mathbf{E} = \text{diag}(GJ_T, EI_2, EI_3).\tag{10.8}$$

Here E and G are Young's modulus and the shear modulus, respectively, while J_T and I_k give a torsional constant and the second moments of area, respectively. The parameters k_j are known as the shear correction factors.

10.3 The weak formulation

A weak formulation of the beam problem stems from the identities

$$\begin{aligned}-\int_0^L \mathbf{N}' \cdot \mathbf{v} \, ds &= \int_0^L \mathbf{F} \cdot \mathbf{v} \, ds, \\ -\int_0^L \mathbf{M}' \cdot \boldsymbol{\psi} \, ds - \int_0^L \mathbf{t} \times \mathbf{N} \cdot \boldsymbol{\psi} \, ds &= \int_0^L \mathbf{G} \cdot \boldsymbol{\psi} \, ds\end{aligned}\tag{10.9}$$

that yield after integration by parts

$$\begin{aligned}\int_0^L \mathbf{N} \cdot \mathbf{v}' \, ds &= \int_0^L \mathbf{F} \cdot \mathbf{v} \, ds + \mathbf{N}(L) \cdot \mathbf{v}(L) - \mathbf{N}(0) \cdot \mathbf{v}(0), \\ \int_0^L \mathbf{M} \cdot \boldsymbol{\psi}' \, ds - \int_0^L \mathbf{t} \times \mathbf{N} \cdot \boldsymbol{\psi} \, ds &= \int_0^L \mathbf{G} \cdot \boldsymbol{\psi} \, ds + \mathbf{M}(L) \cdot \boldsymbol{\psi}(L) - \mathbf{M}(0) \cdot \boldsymbol{\psi}(0).\end{aligned}\tag{10.10}$$

In order to specify forces exerted upon the beam by the environment, we shall set

$$\mathbf{N}_0 = -\mathbf{N}(0) \quad \text{and} \quad \mathbf{N}_L = \mathbf{N}(L).\tag{10.11}$$

Similarly the bending moments applied to the beam are specified as

$$M_0 = -M(0) \quad \text{and} \quad M_L = M(L). \quad (10.12)$$

By using (10.4) and (10.6) and by including transient inertia terms, the equations (10.10) lead to the standard abstraction of the problem: Find $\mathbf{w} \equiv (\mathbf{u}(\cdot, t), \boldsymbol{\theta}(\cdot, t)) \in \mathbf{U}$ such that

$$a(\mathbf{w}, \mathbf{z}) = l(\mathbf{z}), \quad \forall \mathbf{z} \equiv (\mathbf{v}, \boldsymbol{\psi}) \in \mathbf{V} \quad (10.13)$$

with

$$a(\mathbf{w}, \mathbf{z}) = \int_0^L m \ddot{\mathbf{u}} \cdot \mathbf{v} \, ds + \int_0^L \mathbf{I}_m \ddot{\boldsymbol{\theta}} \cdot \boldsymbol{\psi} \, ds + \int_0^L \mathbf{E} \boldsymbol{\theta}' \cdot \boldsymbol{\psi}' \, ds + \int_0^L \mathbf{D}(\mathbf{u}' - \boldsymbol{\theta} \times \mathbf{t}) \cdot (\mathbf{v}' - \boldsymbol{\psi} \times \mathbf{t}) \, ds \quad (10.14)$$

and

$$l(\mathbf{z}) = \int_0^L \mathbf{F} \cdot \mathbf{v} \, ds + \int_0^L \mathbf{G} \cdot \boldsymbol{\psi} \, ds + \mathbf{N}_L \cdot \mathbf{v}(L) + \mathbf{N}_0 \cdot \mathbf{v}(0) + \mathbf{M}_L \cdot \boldsymbol{\psi}(L) + \mathbf{M}_0 \cdot \boldsymbol{\psi}(0). \quad (10.15)$$

Here m is the mass per unit length and the diagonal \mathbf{I}_m gives the (mass) moments of inertia. In addition, \mathbf{U} and \mathbf{V} denote the sets of kinematically admissible functions and test functions, respectively.

10.4 Keywords

Simulation

Coordinate System String Cartesian 3D

The coordinate system should be selected to be three-dimensional, as no specific orientation of the beam axis is supposed.

Material mat id

The following keywords relate to specifying the material parameters and the cross section data.

Youngs Modulus Real

This keyword specifies the value of Young's modulus E .

Shear Modulus Real

This keyword specifies the value of G .

Density Real

This keyword is used for defining the density of the material. The density is needed in transient cases to include the effects of inertial forces.

Cross Section Area Real

This keyword specifies the area A of the cross section.

Principal Direction 2(3) Real

The three components given by using this keyword should define the direction of the local y_2 -axis, so that the vector \mathbf{e}_2 can be computed. This direction must be orthogonal to the beam axis that is determined by the coordinates of the element nodes. If this keyword is not specified, the default value $\mathbf{e}_2 = (0, 0, -1)$ is used.

Torsional Constant Real

To get an approximation of the torsional effects, the value of the parameter J_T can be given.

Second Moment of Area 2 Real

This keyword should give the value of the integral $I_2 \equiv \int_A y_3^2 \, dA$.

Second Moment of Area 3 Real

This keyword should give the value of the integral $I_3 \equiv \int_A y_2^2 \, dA$.

Solver solver id

Equation String

A describing name for the solver.

Procedure File "BeamSolver3D" "TimoshenkoSolver"

The name of the solver subroutine.

Variable String Deflection[U:3 Theta:3]

The name of the solver variable can be chosen freely (but it must be used consistently elsewhere).

Variable DOFs Integer 6

There is no need for using this keyword as the only possible value is 6 and it is given automatically by the solver. The first three components of the solver variable define the displacement \mathbf{u} of the beam axis (the default variable name U), while the rest give the vector $\boldsymbol{\theta}$ (the default variable name Theta). In this connection the components of both the vectors are defined with respect to the global coordinate frame.

Body Force bf id

Body Force k Real

The value of this keyword gives the k th component of the applied force \mathbf{F} with respect to the global coordinate frame.

Boundary Condition bc id

The Dirichlet conditions for the components of \mathbf{u} and $\boldsymbol{\theta}$ can be given in the standard manner. Note that here the components of both vectors are defined with respect to the global coordinate frame.

U i Real

If the default variable name is used, then, with $i=1, 2, 3$, Dirichlet BCs for the components of the displacement \mathbf{u} can be given.

Theta i Real

If the default variable name is used, then, with $i=1, 2, 3$, Dirichlet BCs for the components of the rotation $\boldsymbol{\theta}$ can be given.

Part III

Models of Acoustics

Model 11

The Helmholtz Model

Module name: HelmholtzSolve

Module subroutines: HelmholtzSolver

Module authors: Juha Ruokolainen, Mikko Lyly, Mika Malinen, Peter Råback

Document authors: Juha Ruokolainen, Peter Råback

Document created: 30.3.2006

Document edited: 30.6.2011

11.1 Introduction

This module solves the Helmholtz equation, which is the Fourier transform of the wave equation. In addition to the basic equation the solver may take into consideration variable density, background convections field, simple damping and special boundary conditions with other time-harmonic solvers.

11.2 Theory

For example, sound propagation in air is fairly well described by the wave equation:

$$\frac{1}{c^2} \frac{\partial^2 p}{\partial t^2} - \nabla^2 p = 0. \quad (11.1)$$

When linear the equation may be written in frequency space as

$$k^2 P + \nabla^2 P = 0, \quad (11.2)$$

where $k = \omega/c$. This is the Helmholtz equation. The instantaneous pressure may be computed from the given field P :

$$p(t) = \Re(P e^{i\omega t}) = \Re(P) \cos(\omega t) - \Im(P) \sin(\omega t), \quad (11.3)$$

where $i = \sqrt{-1}$ is the imaginary unity.

In Elmer the equation has an added term which is proportional to first time derivative of the field, whereupon the equation becomes

$$(k^2 - ikD)P + \nabla^2 P = 0, \quad (11.4)$$

where D is the damping factor.

11.2.1 Boundary Conditions

The usual boundary condition for the Helmholtz equation is to give the flux on the boundary:

$$\nabla P \cdot \vec{n} = g, \quad (11.5)$$

also Dirichlet boundary conditions may be set. The Sommerfeldt or far field boundary condition is as follows

$$\nabla P \cdot \vec{n} + \frac{i\omega}{Z} P = 0, \quad (11.6)$$

where the complex-valued quantity Z may be defined by the user. It is noted that incoming and outgoing waves may be approximated by setting $Z = \pm c$, respectively.

A special kind of flux condition is one with a given harmonic velocity field that is obtained from a harmonic solution of a flow or structure equation. When the velocity field \vec{v} is given then the flux is obtained from

$$g = i\omega\rho\vec{v} \cdot \vec{n} \quad (11.7)$$

where ρ is the fluid density. If harmonic displacement is given instead a further term $i\omega$ appears in the equation.

11.3 Keywords

Simulation

This section gives values to parameters concerning the simulation as whole.

Frequency Real

Give simulation frequency in units of 1/s. Alternatively use the Angular Frequency keyword.

Angular Frequency Real

Give simulation frequency in units of 1/rad. Alternatively use the Frequency keyword.

Solver solver id

Note that all the keywords related to linear solver (starting with Linear System) may be used in this solver as well. They are defined elsewhere. Note also that for the Helmholtz equation ILUT preconditioning works well.

Equation String [Helmholtz]

The name of the equation.

Procedure File ["HelmholtzSolve" "HelmholtzSolver"]

This keyword is used to give the Elmer solver the place where to search for the Helmholtz equation solver.

Variable String [Pressure]

Give a name to the field variable.

Variable DOFs Integer [2]

This keyword must be present, and *must* be set to the value 2.

Bubbles Logical

If set to True this keyword activates the bubble stabilization.

Use Density Logical

Historically the solver was able to solve only cases with constant density when it may be eliminated. If the density is however not constant this flag must be set True.

Velocity Variable Name String

If there is a Flow Interface then the name of the harmonic velocity variable may be specified. The default is Flow. Note that normal real valued velocity field is not suitable.

Displacement Variable Name String

If there is a Structure Interface then the name of the harmonic displacement variable may be specified. The default is Displacement. Note that normal real valued displacement field is not suitable, its complex valued eigenmode however is.

Displacement Variable Eigenmode Integer

If eigenmode is used for the interface this keyword is used to specify the number of the mode.

Displacement Variable Frequency Logical

If eigenmode is used for the interface this keyword may be used to choose the frequency to be the frequency of the computed eigenmode.

Equation eq id

The equation section is used to define a set of equations for a body or set of bodies:

Helmholtz Logical

If set to True, solve the Helmholtz equation, the name of the variable must match the Equation setting in the Solver section. Alternatively use the Active Solvers keyword.

Initial Condition ic id

The initial condition section may be used to set initial values for the field variables. The following variables are active:

Pressure i Real

For each the real and imaginary parts of the solved field $i = 1, 2$.

Material mat id

The material section is used to give the material parameter values. The following material parameters may be set in Helmholtz equation.

Sound Speed Real

This keyword is used to give the value of the speed of sound.

Sound Damping Real

This keyword is used to give the value of the damping factor D in equation 11.4.

Density Real

If sound density is varying the density must be specified and its use must be enforced by the Use Density keyword.

Convection Velocity i Real

If the pressure field is convected by a background velocity field (as in the Doppler effect) then this keyword is used to give the velocity field.

Body Force bf id

Pressure Source i Real

The pressure sources of the real ($i = 1$) and complex ($i = 2$) parts. The use of this is rather seldom.

Boundary Condition bc id

The boundary condition section holds the parameter values for various boundary condition types. Dirichlet boundary conditions may be set for all the primary field variables. The one related to Helmholtz equations are

Pressure i Real

Dirichlet boundary condition for real and imaginary parts of the variable. Here the values $i = 1, 2$ correspond to the real and imaginary parts of the unknown field.

Wave Flux 1,2 Real

Real and imaginary parts of the boundary flux. Here the values $i = 1, 2$ correspond to the real and imaginary parts of the boundary flux.

Wave Impedance 1,2 Real

This keyword may be used to define the real and imaginary parts of the quantity Z in (11.6). Here the values $i = 1, 2$ correspond to the real and imaginary parts of Z .

Flow Interface Logical

Use harmonic velocity field to set the flux.

Structure Interface Logical

Use harmonic displacement field to set the flux.

Model 12

The Linearized Navier–Stokes Equations in the Frequency Domain

Module name: Acoustics

Module subroutines: AcousticsSolver

Module authors: Mika Malinen

Document authors: Mika Malinen

Document edited: Aug 14, 2012

12.1 Introduction

The basic acoustic equations such as the Helmholtz equation, which is frequently taken as the starting point in acoustic analyses, are based on the assumption of lossless flow, i.e. the effects of viscosity and heat conduction are neglected. These effects are significant, however, in thin zones near a solid boundary. In this chapter, a system of acoustic field equations taking into account the effects of viscosity and heat conduction is described. Consideration is confined to the time-harmonic solution of these equations.

12.2 Mathematical model

The acoustic field equations may be derived using the general principles of continuum mechanics and supplementing these equations by suitable constitutive equations applicable for the fluid flow. Here the linearized versions of such equations are used to derive an approximate system of field equations appropriate to the small-amplitude acoustics problem.

In the following the velocity, density, pressure and temperature fields associated with the flow are denoted by \vec{v} , ρ , p and T , respectively. The notations ρ_0 , p_0 and T_0 are used for the values of the density, pressure and temperature at the equilibrium state.

12.2.1 The field equations

Consider the acoustic equations based on the linearized equation of motion, the constitutive equation relating the stress to the motion for a Newtonian fluid, the kinematic relation, the linearized continuity equation and

the linearized energy equation

$$\begin{aligned}
 \rho_0 \frac{\partial \vec{v}}{\partial t} &= \nabla \cdot \bar{\bar{\sigma}} + \rho_0 \vec{b}, \\
 \bar{\bar{\sigma}} &= -p \bar{\bar{I}} + \lambda (\nabla \cdot \vec{v}) \bar{\bar{I}} + 2\mu \bar{\bar{D}}(\vec{v}), \\
 \bar{\bar{D}}(\vec{v}) &= \frac{1}{2} (\nabla \vec{v} + \nabla \vec{v}^T), \\
 \frac{\partial \rho}{\partial t} &= -\rho_0 \nabla \cdot \vec{v}, \\
 \rho_0 \frac{du}{dt} &= \kappa \Delta T - p_0 \nabla \cdot \vec{v} + \rho_0 h.
 \end{aligned} \tag{12.1}$$

Here $\bar{\bar{\sigma}}$ is the stress tensor, \vec{b} is the body force (per unit mass), λ and μ are parameters characterizing the viscosity of the fluid, u is the specific internal energy, κ is the heat conductivity and h is the internal supply of heat.

We supplement the system (12.1) by suitable equations of state assuming that the properties of the medium are expressible as functions of two state variables, say the temperature and density. We denote the specific entropy (entropy per unit mass) and its equilibrium value by s and s_0 and assume that the relation

$$du = T_0 ds + (p_0/\rho_0^2) d\rho \tag{12.2}$$

is valid. In addition, we approximate the equations which give the changes of pressure and specific entropy in terms of the changes of the state variables by

$$p - p_0 = \frac{(\gamma - 1)\rho_0 C_V}{T_0 \beta} (T - T_0) + \frac{(\gamma - 1)C_V}{T_0 \beta^2} (\rho - \rho_0) \tag{12.3}$$

and

$$s - s_0 = \frac{C_V}{T_0} (T - T_0) - \frac{C_V(\gamma - 1)}{T_0 \rho_0 \beta} (\rho - \rho_0), \tag{12.4}$$

where C_V is the specific heat at constant volume (per unit mass), γ is the ratio of the specific heats at constant pressure and constant volume and β is the coefficient of thermal expansion defined by

$$\beta = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p. \tag{12.5}$$

Confining consideration to the time-harmonic case, the solutions of the primary unknowns are assumed to be of the form

$$\begin{aligned}
 \vec{v}(x, t) &= \vec{v}(x) \exp(i\omega t), \\
 \rho(x, t) &= \rho_0 + \rho(x) \exp(i\omega t), \\
 T(x, t) &= T_0 + T(x) \exp(i\omega t),
 \end{aligned} \tag{12.6}$$

where ω is the angular frequency. By the substitution of (12.6), the system of field equations based on (12.1)–(12.4) may be reduced to a system where the only unknown fields are the amplitudes $\vec{v}(x)$ and $T(x)$ of the disturbances of the velocity and temperature fields. The reduced system may be written as

$$\begin{aligned}
 i\omega \rho_0 \vec{v} + \frac{(\gamma - 1)C_V \rho_0}{\beta T_0} \nabla T - \left(\lambda + \mu - \frac{i(\gamma - 1)C_V \rho_0}{\omega T_0 \beta^2} \right) \nabla (\nabla \cdot \vec{v}) - \mu \Delta \vec{v} &= \rho_0 \vec{b}, \\
 -\kappa \Delta T + i\omega \rho_0 C_V T + \frac{(\gamma - 1)C_V \rho_0}{\beta} \nabla \cdot \vec{v} &= \rho_0 h.
 \end{aligned} \tag{12.7}$$

It is noted that after the solution of the velocity and temperature the amplitudes $p(x)$ and $\rho(x)$ of the disturbances of the pressure and density fields can readily be obtained from the relations

$$\begin{aligned}
 p &= \frac{(\gamma - 1)C_V \rho_0}{\beta T_0} \left(T + \frac{i}{\omega \beta} \nabla \cdot \vec{v} \right), \\
 \rho &= \frac{i\rho_0}{\omega} \nabla \cdot \vec{v}.
 \end{aligned} \tag{12.8}$$

For numerical approximation the system (12.7) is rewritten as a mixed problem; to motivate this, see [2]. The mixed formulation is written as

$$\begin{aligned} \vec{v} - i\nabla\tau - i\nabla\phi + i\epsilon\nabla(\nabla \cdot \vec{v}) + i\epsilon\Delta\vec{v} &= -(i/\omega)\vec{b}, \\ -\frac{i\epsilon}{(\gamma-1)\delta}\Delta\tau - \frac{1}{\gamma-1}\tau + \frac{1}{1+i\gamma k^2\epsilon\eta}\phi &= \frac{ih}{\beta T_0\omega^2}, \\ i\nabla \cdot \vec{v} - \frac{\gamma k^2}{1+i\gamma k^2\epsilon\eta}\phi &= 0, \end{aligned} \quad (12.9)$$

where ϕ is an auxiliary unknown, τ is the scaled temperature defined by

$$\tau = \frac{\omega\beta}{\gamma k^2} T \quad (12.10)$$

and

$$k = \frac{\omega}{c} \quad \epsilon = \frac{\mu}{\rho_0\omega} \quad \delta = \frac{C_V\mu}{\kappa} \quad \eta = \frac{\lambda}{\mu} \quad (12.11)$$

with c the adiabatic sound speed defined by the relation

$$T_0\beta^2 c^2 = \gamma(\gamma-1)C_V. \quad (12.12)$$

It should be noted that although the solver of the acoustic equations is based on the formulation (12.9), the solver overwrites the approximations of τ and ϕ by the unscaled temperature and the pressure, which may be expressed as

$$p = \rho_0\omega\left(\tau + \frac{1}{1+i\gamma k^2\epsilon\eta}\phi\right). \quad (12.13)$$

It is assumed that $\beta = 1/T_0$. This value is obtained by evaluating the coefficient of thermal expansion for the equilibrium values of the state variables in the case of an ideal gas.

12.2.2 Boundary conditions

Suitable boundary conditions must be adjoined to the field equations (12.1). In a usual manner, one may specify any component of the velocity vector on the boundary. Alternatively, if the component of the velocity vector is not specified at a point on the boundary, the corresponding component of the surface force vector may be prescribed. Similarly, as a boundary condition for the energy equation one may specify either the disturbance of the temperature or zero heat flux (the default boundary condition) on the boundary.

Specifying two impedances on the boundary provides an alternative way of prescribing boundary conditions in the normal direction to the boundary. Firstly, one may specify the specific acoustic impedance Z which is defined to be the ratio of the normal component of the surface force vector (which equals to the pressure in the case of a nonviscous Newtonian fluid with no bulk viscosity) to the normal component of the velocity vector at a point on the boundary, i.e. one may specify

$$Z = \frac{\vec{n} \cdot \vec{\sigma} \vec{n}}{\vec{v} \cdot \vec{n}},$$

where \vec{n} is the outward unit normal vector to the boundary. Secondly, one may prescribe the ratio of the heat flux to the disturbance of the temperature at a point on the boundary by specifying

$$Z_T = \frac{\nabla T(x) \cdot \vec{n}}{T(x)}.$$

For example, outgoing waves may be approximated by setting $Z = -\rho_0 c$ and $Z_T = -i\omega/c$ on the outflow boundary.

Slip boundary conditions may also be used. The velocity slip boundary condition relating the tangential component of the surface force vector to the tangential velocity jump at a point on the boundary is written in the form

$$\vec{\sigma} \vec{n} \cdot \vec{t} = -\frac{c_\sigma}{2-c_\sigma} \left(\frac{2(\gamma-1)C_V(T_0+T_w)}{\pi} \right)^{1/2} \rho_0 (\vec{v} \cdot \vec{t} - \vec{v}_w \cdot \vec{t}),$$

where \vec{t} is a tangent vector to the boundary, c_σ is the momentum accommodation coefficient and T_w and \vec{v}_w are the reference wall temperature and velocity. Here the reference wall temperature is defined to be the deviation of the wall temperature from the equilibrium temperature T_0 . The similar boundary condition for the heat flux is given by

$$\kappa \nabla T \cdot \vec{n} = -\frac{c_T(\gamma + 1)}{2(2 - c_T)} \left(\frac{2(\gamma - 1)C_V(T_0 + T_w)}{\pi} \right)^{1/2} \rho_0 C_V (T - T_w),$$

where c_T is the energy accommodation coefficient.

12.3 The use of block preconditioning

The finite element approximation of the system (12.9) leads usually to large linear systems which have to be solved using preconditioned iterative methods. The general preconditioners available in Elmer may not always work satisfactorily well when the size of the system becomes larger and larger. To facilitate the solution of large problems, a problem-specific strategy for solving the linear systems that arise from the discretization of (12.9) has been developed. We describe the essential features of this solution method in this section; for a full description see [1].

The solution strategy discussed here is based on using nested GCR iterations in combination with a special block-preconditioner. Given the linear system

$$KU = F$$

the standard GCR method generates a sequence of improving approximations such that each iterate $U^{(k)}$ minimizes $\|F - KU^{(k)}\|$ over the so-called Krylov subspace. The standard algorithm can be modified easily so that the update direction can be chosen flexibly. Obviously, an optimal update direction would be given by the current error $e^{(k)} = U - U^{(k)}$. To find an approximation to the error one may apply an iterative method to

$$Ke^{(k)} = r^{(k)},$$

where $r^{(k)} = F - KU^{(k)}$ is the residual. The preconditioned GCR algorithm which employs this idea to find the update direction can be described as follows:

Form an initial guess $U^{(0)}$

$$r^{(0)} = F - KU^{(0)}$$

$$k = 0$$

while (Stopping criterion is not met)

Solve $Ke^{(k)} = r^{(k)}$ iteratively using at most m iteration steps

$$v^{(k+1)} = Ke^{(k)}$$

do $j = 1, k$

$$v^{(k+1)} = v^{(k+1)} - \langle v^{(j)}, v^{(k+1)} \rangle v^{(j)}$$

$$s^{(k+1)} = s^{(k+1)} - \langle v^{(j)}, s^{(k+1)} \rangle s^{(j)}$$

end do

$$v^{(k+1)} = v^{(k+1)} / \|v^{(k+1)}\|$$

$$s^{(k+1)} = s^{(k+1)} / \|v^{(k+1)}\|$$

$$U^{(k+1)} = U^{(k)} + \langle v^{(k+1)}, r^{(k)} \rangle s^{(k+1)}$$

$$r^{(k+1)} = r^{(k)} - \langle v^{(k+1)}, r^{(k)} \rangle v^{(k+1)}$$

$$k = k + 1$$

end while

Here the inner product and norm are defined by $\langle v, r \rangle = \bar{v} \cdot r$ and $\|v\| = \langle v, v \rangle^{1/2}$. The GCR iteration steps used to update the approximation of U are referred to as outer iterations, while the iteration steps of the preconditioning iterative method used for solving the new search direction $s^{(k+1)}$ are referred to as inner iterations.

Here the GCR algorithm is also used as the inner iterative method. In connection with the inner iterations a special block-preconditioner is used. The preconditioning is done by solving approximately the block-triangular system of the form

$$\begin{bmatrix} A & B^* & B^* & 0 \\ 0 & C & D & 0 \\ 0 & E & G & H \\ 0 & 0 & 0 & M \end{bmatrix} \begin{bmatrix} s_v \\ s_\tau \\ s_\phi \\ \psi \end{bmatrix} = \begin{bmatrix} r_v \\ r_\tau \\ r_\phi \\ r_\phi \end{bmatrix}, \quad (12.14)$$

where s_v , s_τ and s_ϕ are update directions for the errors of v , τ and ϕ . In addition, ψ is an auxiliary unknown which has been introduced so as to handle the boundary conditions of the preconditioner in a consistent way. In practice, an approximate solution of (12.14) is constructed by applying iterative methods to the systems of the type

$$M\psi = r_\phi, \quad (12.15)$$

$$\begin{bmatrix} C & D \\ E & G \end{bmatrix} \begin{bmatrix} s_\tau \\ s_\phi \end{bmatrix} = \begin{bmatrix} r_\tau \\ \hat{r}_\phi \end{bmatrix} \quad (12.16)$$

and

$$As_v = \hat{r}_v, \quad (12.17)$$

where \hat{r}_ϕ and \hat{r}_v are modified right-hand sides the computation of which requires the evaluation of certain matrix-vector products. The special solver discussed hence requires that iterations are performed on three levels.

One of the key ideas in the nested application of the GCR algorithm is that the outer iteration can be made rapidly convergent. Consequently the optimality of the outer iteration need not be sacrificed by using such techniques as restarting or truncation. A few inner iterations are usually enough to produce a useful reduction in the outer iteration residual. Therefore the maximum number of iterations the inner iterative method may take need not be large. We have found that limiting the number of inner iterations by taking $m = 5$ (this is the default value) or $m = 10$ leads often to an efficient method. In addition to specifying the maximum number of inner iterations, the user can control the residual reduction in the outer iteration process by specifying the error tolerance δ_{inner} so that the inner GCR iteration is stopped if

$$\|r^{(k)} - K\hat{s}^{(k+1)}\| < \delta_{\text{inner}}\|r^{(k)}\|, \quad (12.18)$$

where $\hat{s}^{(k+1)}$ is the approximation to $s^{(k+1)}$. The default value of δ_{inner} is 0.1.

Ideally a mild stopping criterion should be used in the solution of the linear systems of the type (12.15)–(12.17) which arise in the block-preconditioning of the inner iteration. The iterative solution of (12.15) being a cheap operation, the overall cost of the block-preconditioning is essentially determined by the solution of the systems of the type (12.16) and (12.17). These systems are solved using the preconditioned BiCGStab(l) method. In this connection the Jacobi and incomplete LU factorization preconditioners can be applied.

12.4 Utilities

The dissipative acoustics solver may be used in resolving the acoustic impedance of a system. The value of the impedance defined by

$$z_i = \frac{\int_{S_i} p \, dS}{\int_{S_i} \vec{v} \cdot (-\vec{n}) \, dS} \quad (12.19)$$

may automatically be calculated for a given boundary S_i . Here this impedance will be referred to as the specific acoustic impedance of the surface (S_i).

The acoustic impedance is divided into two parts, a part in phase with velocity and a part out of phase with velocity. The value of the impedance z_i is meaningful only when the velocity on the input boundary is considered. It is though possible to calculate the response over an other boundary S_j and to compare it to the input velocity, i.e. one may compute

$$z_{ij} = \frac{\int_{S_j} p \, dS}{\int_{S_i} \vec{v} \cdot (-\vec{n}) \, dS}. \quad (12.20)$$

This impedance is here called the cross specific acoustic impedance.

12.5 Keywords

The following keywords are particularly related to the acoustics solver.

Simulation

Angular Frequency Real

This keyword is used to declare the angular frequency. Alternatively one may define the frequency by using the `Frequency` keyword.

Frequency Real

This keyword is used to declare the frequency. Alternatively one may define the angular frequency by using the `Angular Frequency` keyword.

Simulation Type String

The value of this keyword should be either `Steady State` or `Scanning`. The value `Scanning` may be used to obtain results for several frequencies by using a single `sif`-file.

Coordinate System String

The coordinate system must be set to be one of the following options: `Cartesian 2D`, `Cartesian 3D` or `Axi Symmetric`.

Solver solver-id

The following keywords may be used in the solver section that contains solver parameters for the acoustics solver.

Equation String

This keyword can be used to give a name for the discrete acoustic equations

Procedure File Acoustics AcousticsSolver

This keyword is used to give the Elmer solver the place where to search for the acoustics solver.

Variable String Flow

The name `Flow` is used for the solution of the acoustics equations consisting of the amplitudes of the disturbances of the velocity, temperature and pressure from the equilibrium state (note that the disturbance of the density is not computed explicitly). The acoustics solver uses a convention that if dim is the coordinate system dimension then the components $1, \dots, 2 \times dim$ of `Flow` give the real and imaginary parts of velocities (`Flow.1` and `Flow.2` are the real and imaginary parts of the first velocity component, etc.). The temperature and pressure solutions come after the velocity solution.

Variable Dofs Integer

The value of this keyword should equal to $2 \times (dim + 2)$ where dim is the coordinate system dimension.

Element String

The use of standard finite elements in the approximation of the acoustic equations is likely to lead to an unstable method. The finite element formulation can be stabilised by using additional bubble finite element functions in the approximation of velocities. If this keyword is given the value `p:1 b:n`, with n an integer, then n additional bubble functions contained in the p -element library are used in the approximation of each velocity component.

Bubbles in Global System Logical

This keyword should be given the value `False`, so that the additional bubble basis functions needed for the stability are eliminated via the static condensation.

Utilize Previous Solution Logical

If a single `sif`-file is used to compute the solutions for several frequencies, then the previous solution can be used as an initial guess for the next iterative solution. This can be done by giving the value `True` for this keyword.

Material material-id

Specific Heat Real

This keyword is used to define the specific heat (per unit mass) at constant volume.

Specific Heat Ratio Real

This keyword is used to define the ratio of the specific heats at constant pressure and constant volume.

Equilibrium Density Real

This keyword is used to declare the density at the equilibrium state.

Equilibrium Temperature Real

This keyword is used to declare the absolute temperature at the equilibrium state.

Heat Conductivity Real

This keyword is used to define the value of the heat conductivity.

Viscosity Real

This keyword is used to define the value of the viscosity μ .

Bulk Viscosity Real

The material parameter λ is determined by giving the bulk viscosity κ' defined by $\kappa' = \lambda + 2/3\mu$. If the value of this keyword is not given, the Stokes condition is assumed, i.e. the value of λ is determined by the condition $\kappa' = 0$.

Re Heat Source Real

This keyword is used to define the real part of the heat source (per unit mass).

Im Heat Source Real

This keyword is used to define the imaginary part of the heat source (per unit mass).

Re Body Force i Real

This keyword is used to define the real part of the i 's component of the body force vector (per unit mass).

Im Body Force i Real

This keyword is used to define the imaginary part of the i 's component of the body force vector (per unit mass).

Boundary Condition bc-id

Re Velocity i Real

This keyword is used to prescribe the real part of the i 's component of the velocity vector.

Im Velocity i Real

This keyword is used to prescribe the imaginary part of the i 's component of the velocity vector.

Re Temperature Real

This keyword is used to prescribe the real part of the amplitude of the disturbance of temperature.

Im Temperature Real

This keyword is used to prescribe the imaginary part of the amplitude of the disturbance of temperature.

Re Surface Traction i Real

This keyword is used to define the real part of the i 's component of the surface force vector.

Im Surface Traction i Real

This keyword is used to define the imaginary part of the i 's component of the surface force vector.

Re Specific Acoustic Impedance Real

This keyword is used to define the real part of the ratio of the normal component of the surface force vector to the normal component of the velocity vector at a point on the boundary.

Im Specific Acoustic Impedance Real

This keyword is used to define the imaginary part of the ratio of the normal component of the surface force vector to the normal component of the velocity vector at a point on the boundary.

Re Specific Thermal Impedance Real

This keyword is used to define the real part of the ratio of the normal derivative of temperature to the disturbance of the temperature at a point on the boundary.

Im Specific Thermal Impedance Real

This keyword is used to define the imaginary part of the ratio of the normal derivative of temperature to the disturbance of the temperature at a point on the boundary.

Slip Boundary Logical

The value of this keyword should be set to be True if slip boundary conditions were given.

Momentum Accommodation Coefficient Real

This keyword is used to define the momentum accommodation coefficient c_σ .

Energy Accommodation Coefficient Real

This keyword is used to define the energy accommodation coefficient c_T .

Re Reference Wall Velocity i Real

This keyword is used to prescribe the real part of the i 's component of the reference wall velocity.

Im Reference Wall Velocity i Real

This keyword is used to prescribe the imaginary part of the i 's component of the reference wall velocity.

Reference Wall Temperature Real

This keyword is used to define the reference wall temperature.

Calculate Acoustic Impedance Logical

This keyword is used to define the boundary for which the specific acoustic impedance z_i is calculated.

Impedance Target Boundary Logical

When calculating the cross impedance z_{ij} , this keyword defines the boundary S_j . The input velocity boundary (S_i) is defined using the Calculate Acoustic Impedance keyword.

The following keywords are related to the use of block preconditioning and may be given in the Solver section.

Solver solver-id

Block Preconditioning Logical

The value of this keyword should be set to be True to enable the use of block preconditioning.

Max Outer Iterations Integer

The value of this keyword defines the maximum number of outer iterations.

Max Inner GCR Iterations Integer

This keyword is used to define the value of the parameter m , i.e. the maximum number of inner iterations. The default value is 5.

Ratio of Convergence Tolerances Real

This keyword is used to define the stopping criterion for the outer iteration. The outer iteration is stopped when

$$\|F - KU^{(k)}\|_\infty < (\varepsilon_r \times \varepsilon)(\|K\|_\infty \|U^{(k)}\|_\infty + \|F\|_\infty).$$

Here ε_r is defined using this keyword and ε is the value of the Linear System Convergence Tolerance keyword. Having $\varepsilon_r \ll 1$ is desirable.

Residual Reduction Ratio Real

This keyword is used to define the value of the parameter δ_{inner} in the stopping criterion (12.18). The default value is 0.1.

Linear System Convergence Tolerance *Real*

In connection with the block-preconditioning the keyword **Linear System Convergence Tolerance** defines the stopping criterion used in connection with the iterative solution of (12.16) and (12.17). In this connection the stopping criterion of the type

$$\|\hat{r}_v^{(k)} - As_v^{(k)}\| < \varepsilon \|\hat{r}_v^{(k)}\|$$

is used. Here ε is the value of this keyword. It is noted that the solution accuracy of (12.15) need not be specified by the user.

Velocity Convergence Tolerance *Real*

The systems (12.16) and (12.17) may solved with different degrees of accuracy. Instead of using the **Linear System Convergence Tolerance** keyword one may specify the solution accuracy for (12.17) by using this keyword.

Schur Complement Convergence Tolerance *Real*

The systems (12.16) and (12.17) may solved with different degrees of accuracy. Instead of using the **Linear System Convergence Tolerance** keyword one may specify the solution accuracy for (12.16) by using this keyword.

Linear System Max Iterations *Integer*

In connection with the block-preconditioning the **Linear System Max Iterations** keyword is used for defining the maximum number of iteration steps which can be taken in the iterative solution of (12.15)–(12.17).

Velocity Assembly *Logical*

The coefficient matrix A in (12.17) corresponds to the (1,1) block of the coefficient matrix K . As the elements of A can be extracted from K , the assembly of A can be avoided if a diagonal preconditioning is used in the iterative solution of (12.17). If an incomplete factorization preconditioner is used, the matrix A is assembled explicitly. In this case the value **True** must be given for this keyword.

ILU Order for Schur Complement *Integer*

The value of this keyword defines the fill level for the incomplete LU factorization preconditioner that is applied in the iterative solution of the linear systems of the type (12.16).

ILU Order for Velocities *Integer*

The value of this keyword defines the fill level for the incomplete LU factorization preconditioner that is applied in the iterative solution of the linear systems of the type (12.17). If this keyword is not given, then a diagonal preconditioning is used. This keyword has an effect only when the **Velocity Assembly** keyword is given the value **True**.

Bibliography

- [1] M. Malinen. Boundary conditions in the schur complement preconditioning of dissipative acoustic equations. *SIAM J. Sci. Comput.*, 29:1567–1592, 2007.
- [2] M. Malinen, M. Lyly, P. Råback, A. Kärkkäinen, and L. Kärkkäinen. A finite element method for the modeling of thermo-viscous effects in acoustics. In P. Neittaanmäki et. al., editor, *Proceedings of the 4th European Congress on Computational Methods in Applied Sciences and Engineering*, 2004.

Model 13

Wave Equation

Module name: WaveSolver

Module subroutines: WaveSolver

Module authors: Juha Ruokolainen, Peter Råback, Mika Malinen

Document authors: Mika Malinen

Document created: April 19, 2018

Document edited: April 19, 2018

13.1 Introduction

This module can be used to solve a generalized version of the wave equation in time domain. It provides an alternative to the Helmholtz equation which follows from the wave equation by transforming to the frequency domain.

13.2 Theory

This solver can be used to handle the following generalized version of the wave equation:

$$-\frac{1}{c^2} \frac{\partial^2 p}{\partial t^2} + \Delta p + \frac{\eta}{c^2} \Delta \left(\frac{\partial p}{\partial t} \right) - \frac{\alpha}{c^2} \frac{\partial p}{\partial t} = f, \quad (13.1)$$

with c and f being the sound speed and a source function, respectively. In addition, η and α are model parameters, which can also be set to be zero to obtain the standard wave equation.

In the case of sound waves the equation (13.1) can be obtained from the equation of motion and the continuity equation by linearization. By assuming an isentropic fluid and irrotational flow, we then come to considering the velocity-pressure system

$$\begin{aligned} \rho_0 \frac{\partial \mathbf{v}}{\partial t} - \hat{\eta} \nabla (\nabla \cdot \mathbf{v}) + \hat{\alpha} \mathbf{v} - \nabla P &= \rho_0 \mathbf{b}, \\ \frac{1}{\rho_0 c^2} \frac{\partial P}{\partial t} + \nabla \cdot \mathbf{v} &= 0, \end{aligned} \quad (13.2)$$

where ρ_0 is the density of the fluid at the equilibrium state, $\hat{\eta}$ is a parameter related to the fluid viscosity and $\hat{\alpha}$ is a reaction parameter. If we use the continuity equation to eliminate the divergence term from the equation of motion and set $\eta = \hat{\eta}/\rho_0$, $\alpha = \hat{\alpha}/\rho_0$ and $p = P/\rho_0$, the velocity-pressure system can also be expressed as

$$\begin{aligned} \frac{\partial \mathbf{v}}{\partial t} + \frac{\eta}{c^2} \nabla \left(\frac{\partial p}{\partial t} \right) + \alpha \mathbf{v} - \nabla p &= \mathbf{b}, \\ \frac{1}{c^2} \frac{\partial p}{\partial t} + \nabla \cdot \mathbf{v} &= 0. \end{aligned} \quad (13.3)$$

The elimination of the velocity from the time-differentiated continuity equation

$$-\frac{1}{c^2} \frac{\partial^2 p}{\partial t^2} - \nabla \cdot \left(\frac{\partial \mathbf{v}}{\partial t} \right) = 0 \quad (13.4)$$

then yields (13.1), with $f \equiv \nabla \cdot \mathbf{b}$.

Boundary conditions. Basically boundary conditions of three types can be given. In addition to a basic Dirichlet constraint, one can specify the Neumann constraint

$$-\nabla p \cdot \mathbf{n} - \frac{\eta}{c^2} \nabla \left(\frac{\partial p}{\partial t} \right) \cdot \mathbf{n} = g. \quad (13.5)$$

In the absence of body forces and reaction terms ($\mathbf{b} = \mathbf{0}$ and $\alpha = 0$), the value of g is related to the normal component of the acceleration via

$$g = -\frac{\partial \mathbf{v}}{\partial t} \cdot \mathbf{n}.$$

The final option is to set

$$-\nabla p \cdot \mathbf{n} - \frac{\eta}{c^2} \nabla \left(\frac{\partial p}{\partial t} \right) \cdot \mathbf{n} = \frac{1}{c} \frac{\partial p}{\partial t} \quad (13.6)$$

in order to approximate outgoing waves, especially when η is small.

13.3 Keywords

`Solver solver id`

The following keywords are especially related to the wave equation solver.

`Equation String`

This keyword can be used to give a name for the equation handled by the solver.

`Procedure File "WaveSolver" "WaveSolver"`

This specifies the name of the solver subroutine.

`Variable String "Excess Pressure"`

The name of the solver variable can be chosen freely.

`Variable DOFs Integer`

The value of this keyword must be 1.

`Time Derivative Order Integer`

There is no need for using this keyword as the only possible value is 2 and it is given automatically by the solver.

`Initial Condition ic-id`

The initial condition section may be used to set initial values for the field variable. The form of the commands depends on the name given for the solver variable.

`Material mat id`

The material section is used to give the values of the material parameters.

`Sound Speed Real`

This keyword is used to give the value of the sound speed c .

`Sound Damping Real`

This keyword is used to give the value of the parameter η .

`Sound Reaction Damping Real`

This keyword is used to give the value of the parameter α .

`Body Force bf id`

Although volumetric sources may be rare, there is an option to specify such a source.

Sound Source Real

This keyword can be used to specify the source function $f \equiv \nabla \cdot \mathbf{b}$.

Boundary Condition bc id

Special keywords are used to specify Neumann and outflow constraints, while a Dirichlet constraint can be given in a standard manner.

Excess Pressure Real

The form of the command to specify a Dirichlet constraint naturally depends on the name given for the solver variable.

Source Acceleration Real

This keyword can be used to specify the source function g .

Outflow Boundary Logical

This keyword can be used to activate the use of the constraint (13.6). An alternate keyword for the same purpose is Plane Wave BC.

Model 14

Large-amplitude Wave Motion in Air

Module names: CompressibleNS

Module subroutines: CompressibleNS

Module authors: Mika Malinen

Document authors: Mika Malinen

Document edited: Aug 14, 2012

14.1 Introduction

This module contains a monolithic solver for the compressible Navier–Stokes equations subject to the ideal gas law. It can be used to model the fully nonlinear wave propagation in the time domain.

14.2 Mathematical model

The acoustic wave motion in a fluid is generally characterized by the compressional Navier–Stokes equations. If the medium obeys the ideal gas law, so that the fluid pressure p satisfies

$$p = R\rho T, \quad (14.1)$$

the Navier–Stokes system may be reduced to consist of the equation of motion

$$\rho \left[\frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla) \vec{v} \right] - \mu \Delta \vec{v} - (\mu + \lambda) \nabla (\nabla \cdot \vec{v}) + R\rho \nabla T + RT \nabla \rho = \vec{b}, \quad (14.2)$$

the energy equation

$$\rho C_V \left(\frac{\partial T}{\partial t} + \vec{v} \cdot \nabla T \right) - K \Delta T + R\rho T \nabla \cdot \vec{v} = 0, \quad (14.3)$$

and the continuity equation

$$\frac{\partial \rho}{\partial t} + \vec{v} \cdot \nabla \rho + \rho \nabla \cdot \vec{v} = 0. \quad (14.4)$$

Here \vec{v} , ρ and T are the fluid velocity, density and temperature, respectively, and the material properties are expressed in terms of the viscosity parameters μ and λ , the heat conductivity K and the specific heat C_V , with $R = (\gamma - 1)C_V$.

If ρ_0 and T_0 are the equilibrium values of density and temperature, we may then write

$$\rho = \rho_0 + \delta, \quad T = T_0 + \tau, \quad (14.5)$$

so that δ and τ give the disturbances in ρ and T . To solve the coupled system consisting of (14.2)–(14.4) the fully implicit time integration is employed. At each time level a nonlinear iteration is thus applied. Given

nonlinear iterates \vec{v}_k , τ_k and δ_k , new approximations are generated via

$$\begin{aligned} &(\rho_0 + \delta_k)[(\vec{v}_t)_{k+1} + (\vec{v}_k \cdot \nabla)\vec{v}_{k+1}] - \mu\Delta\vec{v}_{k+1} - (\mu + \lambda)\nabla(\nabla \cdot \vec{v}_{k+1}) \\ &+ R(\rho_0 + \delta_k)\nabla\tau_{k+1} + R(T_0 + \tau_k)\nabla\delta_{k+1} = \vec{b}, \\ &(\rho_0 + \delta_k)C_V[(\tau_t)_{k+1} + \vec{v}_k \cdot \nabla\tau_{k+1}] - K\Delta\tau_{k+1} + R(\rho_0 + \delta_k)(T_0 + \tau_k)\nabla \cdot \vec{v}_{k+1} = 0, \\ &(\rho_t)_{k+1} + \vec{v}_k \cdot \nabla\delta_{k+1} + (\rho_0 + \delta_k)\nabla \cdot \vec{v}_{k+1} = 0, \end{aligned} \quad (14.6)$$

with the time derivatives approximated using suitable finite difference schemes. It is recommended that the BDF(2) method is used for the time discretization. It is also noted that the pressure is not approximated directly, so it has to be computed separately using (14.1) and (14.5).

It should be noted that the solver is tailored to the case of the lowest-order continuous temperature and density approximation. To obtain stable finite element solutions the velocity discretization must be enhanced by using elementwise bubble functions or by rising the polynomial order of the velocity approximation. Therefore a special element type definition in the solver input file should be given.

14.3 Keywords

The keywords that are related especially to this solver are described in the following.

Simulation

Coordinate System String

The coordinate system must be set to be one of the following options: Cartesian 2D, Cartesian 3D or Axi Symmetric.

Solver solver id

Equation String

A name to the equation may be given by using this keyword.

Procedure File "CompressibleNS" "CompressibleNS"

This keyword is used to give the Elmer solver the place where to search for the compressible Navier–Stokes solver.

Variable String

A name to the solver variable should be given by using this keyword.

Variable DOFs Integer

The value of this keyword should equal to $dim + 2$ where dim is the coordinate system dimension. The field variables are organized in such a way that the first dim components correspond to the velocity solution and the temperature and density fluctuations come after the velocity.

Element String

The user has to specify what strategy is used for enhancing the velocity approximation by giving the element type definition. If the command `Element = "p:2"` is given, then the velocity is approximated using the shape functions of the second order elements. The element type definition `Element = "p:1 b:1"` can be given to enhance the velocity approximation with one bubble function.

Material mat id

Equilibrium Density Real

The equilibrium density ρ_0 should be specified by using this keyword.

Equilibrium Temperature Real

The equilibrium temperature T_0 should be specified by using this keyword.

Specific Heat Real

The value of this keyword specifies C_V .

Specific Heat Ratio Real

The value of this keyword specifies γ .

Heat Conductivity Real

The heat conductivity K should be defined by using this keyword.

Viscosity Real

The viscosity parameter μ should be defined by using this keyword.

Bulk Viscosity Real

The viscosity parameter λ is taken to be $\lambda = \kappa - 2/3\mu$, with κ the value of this keyword.

Body Force bc id

Body Force i Real

This keyword defines the i 's component of the body force.

Part IV

Models of Electromagnetism

Model 15

Electrostatics

Module name: StatElecSolve

Module subroutines: StatElecSolver

Module authors: Leila Puska, Antti Pursula, Peter Råback

Document authors: Peter Råback, Antti Pursula

Document edited: May 27th 2013

15.1 Introduction

The macroscopic electromagnetic theory is governed by Maxwell's equations. In Elmer it is possible to solve the electrostatic potential in linear dielectric material and in conducting medium. The dielectric case is described in this Chapter. For static currents, refer to Chapter 16. Based on the potential, various field variables as well as physical parameters, such as capacitance, can be calculated.

15.2 Theory

Maxwell's equations are here written as

$$\nabla \cdot \vec{D} = \rho \quad (15.1)$$

$$\nabla \cdot \vec{B} = 0 \quad (15.2)$$

$$\nabla \times \vec{E} = -\frac{\partial \vec{B}}{\partial t} \quad (15.3)$$

$$\nabla \times \vec{H} = \vec{J} + \frac{\partial \vec{D}}{\partial t} \quad (15.4)$$

For linear materials the fields and fluxes are simply related, $\vec{B} = \mu \vec{H}$ and $\vec{D} = \epsilon \vec{E}$, where the permittivity $\epsilon = \epsilon_0 \epsilon_r$ is defined through the permittivity of vacuum ϵ_0 and the relative permittivity of the material ϵ_r .

In a stationary case the electric field may be expressed with a help of an electric scalar potential ϕ ,

$$\vec{E} = -\nabla \phi. \quad (15.5)$$

Assuming linear material law and using the equation (15.1) gives

$$-\nabla \cdot \epsilon \nabla \phi = \rho. \quad (15.6)$$

This is the electrostatic equation for non-conducting media.

The energy density of the field is

$$e = \frac{1}{2} \vec{E} \cdot \vec{D} = \frac{1}{2} \epsilon (\nabla \phi)^2. \quad (15.7)$$

Thus the total energy of the field may be computed from

$$E = \frac{1}{2} \int_{\Omega} \varepsilon (\nabla \phi)^2 d\Omega. \quad (15.8)$$

If there is only one potential difference Φ present then the capacitance C may be computed from

$$C = \frac{2E}{\Phi^2}. \quad (15.9)$$

15.2.1 Boundary Conditions

For electric potential either Dirichlet or Neumann boundary condition can be used. The Dirichlet boundary condition gives the value of the potential on specified boundaries. The Neumann boundary condition is used to give a flux condition on specified boundaries

$$-\varepsilon \nabla \phi \cdot \vec{n} = g. \quad (15.10)$$

The flux may be defined *e.g.* by the surface charge density: $g = \sigma$.

In case there is a object in infinite space it is of course not possible to extent the volume over it. Instead a spherically symmetric approximation may be used. It results to a flux given by

$$g = \varepsilon \phi \frac{\vec{r} \cdot \vec{n}}{r^2}. \quad (15.11)$$

This may be implemented as an additional term to the matrix so that the linear nature of the problem is maintained.

Conductors are often covered by thin oxidation layers which may contain static charges. The effect of these charges can be taken into account by Robin type of boundary condition which combines the fixed potential value on the conductor and the flux condition due to the static charges

$$g = -\frac{\varepsilon_h}{h} \phi + \frac{1}{2} \rho h + \frac{\varepsilon_h}{h} \Phi_0 \quad \text{on the boundary,} \quad (15.12)$$

where ε_h and h are the permittivity and the thickness of the oxidation layer respectively, ρ is the static charge density of the layer, and Φ_0 is the fixed potential on the conductor.

Note that this formulation is valid only for thin layers. For a larger layer a separate body should be added and a source defined for that.

15.2.2 Capacitance matrix

There is a possibility to compute the capacitance matrix. The algorithm takes use of the original matrix A before the initial conditions are set. Now the point charges are given by

$$q = A\phi. \quad (15.13)$$

The induced charges on a body may be computed by summing up the point charges.

If there are n different bodies the boundary conditions are permuted n times so that body i gets a potential unity while others are set to zero potential,

$$C_{ij} = \sum_{\Gamma_j} q. \quad (15.14)$$

The symmetry of the matrix is ensured afterwards by setting

$$C = \frac{1}{2}(C + C^T). \quad (15.15)$$

15.3 Notes on output control

The user can control which derived quantities (from the list of electric field, electric flux, electric energy, surface charge density and capacitance matrix) are calculated.

There are also available two choices of visualization types for the derived quantities. The node values can be calculated by taking the average of the derived values on neighbouring elements (constant weights). This results often in visually good images. The other possible choice is to weight the average with the size of the elements, which is more accurate and should be used when some other variable depends on these derived values. The latter choice is also the default.

15.4 Keywords

Constants

Permittivity Of Vacuum Real [8.8542e-12]

Solver solver id

Equation String Stat Elec Solver

Variable String Potential

This may be of any name as far as it is used consistently also elsewhere.

Variable DOFs Integer 1

Degrees of freedom for the potential.

Procedure File "StatElecSolve" "StatElecSolver"

Following are listed four keywords with default values for output control.

Calculate Electric Field Logical [True]

Calculate Electric Flux Logical [True]

Calculate Electric Energy Logical [False]

Calculate Surface Charge Logical [False]

Calculate Capacitance Matrix Logical [False]

Capacitance Bodies Integer

In case of a capacitance matrix computation the number of bodies at different potential must be given (not accounting the ground).

Capacitance Matrix Filename String

The name of the file where capacitance matrix is being saved. The default is `cmatrix.dat`.

Constant Weights Logical [True]

Used to turn constant weighting on for the results.

Potential Difference Real

Used to give the potential difference for which the capacitance is calculated, when capacitance matrix calculation is not performed. This keyword gives thus the voltage between the electrodes of a simple capacitor. The voltage has to be consistent with the potentials defined in boundary conditions.

Material mat id

Relative Permittivity Real

Body Force bodyforce id

Charge Density Real

Boundary Condition `bc id`

Potential `Real`

If the name of the primary variable is potential then this sets the Dirichlet boundary condition.

Electric Flux `Real`

Neumann boundary condition for g .

Surface Charge Density `Real`

Another way to define flux condition. Identical to the previous keyword.

Electric Infinity BC `Logical`

The spherical approximation for the open boundaries extending to infinity.

The following five keywords are used if a thin oxidation layer is modeled. Note that these are only active if the `Electric Flux BC` keyword is set to `True`.

Layer Thickness `Real`

Defines the thickness of the oxidation layer. This is presumed to extend on the outside the boundary.

Layer Relative Permittivity `Real`

The relative permittivity of the oxidation layer.

Layer Charge Density `Real`

The volume charge density in the oxidation layer.

Electrode Potential `Real`

The potential on the conductor behind the oxidation layer.

Nominal Potential Difference `Real`

The potential difference of the system.

Capacitance Body `Integer i`

These should number from $i=1$ up to `Capacitance Bodies`. The ground may be given directly with zero potential or with value 0 for this keyword. This definition is only needed in the computation of the capacitance matrix where the potential is permuted in a very specific way.

Model 16

Static Current Conduction

Module name: StatCurrentSolve

Module subroutines: StatCurrentSolver

Module authors: Leila Puska, Antti Pursula, Peter Råback

Document authors: Antti Pursula

Document edited: May 27th 2013

16.1 Introduction

The macroscopic electromagnetic theory is governed by the Maxwell's equations. This module solves the electrostatic potential in conducting medium allowing volume currents and electric power loss (Joule heating) to be derived.

16.2 Theory

In the electroquasistatic approximation Maxwell's equations are written as

$$\nabla \cdot \vec{D} = \rho \quad (16.1)$$

$$\nabla \times \vec{E} \simeq 0 \quad (16.2)$$

$$\nabla \times \vec{H} = \vec{J} + \frac{\partial \vec{D}}{\partial t} \quad (16.3)$$

so that the electric field may be expressed in terms of an electric scalar potential ϕ as

$$\vec{E} = -\nabla \phi. \quad (16.4)$$

In addition, the continuity equation for electric charges is easily obtained from (16.1) and (16.3):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \vec{J} = 0. \quad (16.5)$$

The Ohm's law for conducting material gives the relationship between current density and electric field,

$$\vec{J} = \sigma \vec{E} \quad (16.6)$$

where σ is the electric conductivity. Starting from the continuity equation (16.5) and using the equations (16.6) and (16.4) we get

$$-\nabla \cdot \sigma \nabla \phi - \nabla \cdot \epsilon \nabla \frac{d\phi}{dt} = \rho_t. \quad (16.7)$$

This Poisson equation is used to solve the electric potential. The source term is often zero but in some cases it might be necessary.

The volume current density is now calculated by

$$\vec{J} = -\sigma \nabla \phi, \quad (16.8)$$

and electric power loss density which is turned into heat by

$$h = \nabla \phi \cdot \sigma \nabla \phi. \quad (16.9)$$

The latter is often called the Joule heating. The total heating power is found by integrating the above equation over the conducting volume.

The user may also compute the nodal heating which is just the integral of the heating to nodes.

16.2.1 Boundary Conditions

For electric potential either Dirichlet or Neumann boundary condition can be used. The Dirichlet boundary condition gives the value of the potential on specified boundaries. The Neumann boundary condition is used to give a current J_b on specified boundaries

$$J_b = \sigma \nabla \phi \cdot \vec{n}. \quad (16.10)$$

16.2.2 Power and current control

Sometimes the desired power or current of the system is known a priori. The control may be applied to the system. When the electric potential has been computed the heating power may be estimated from

$$P = \int_{\Omega} \nabla \phi \cdot \sigma \nabla \phi d\Omega. \quad (16.11)$$

If there is a potential difference U in the system the effective resistance may also be computed from $R = U^2/P$ and the effective current from $I = P/U$.

The control is achieved by multiplying the potential and all derived fields by a suitable variable. For power control the coefficient is

$$C_P = \sqrt{P_0/P}, \quad (16.12)$$

where P_0 is the desired power. For current control the coefficient is

$$C_I = I_0/I, \quad (16.13)$$

where I_0 is the desired total current.

16.3 Note on output control

The user can control which derived quantities (*i.e.* volume current and Joule heating) are calculated and additionally specify if he/she wants to output also the electric conductivity. The latter is useful when the conductivity depends for example on temperature. This feature is available only for isotropic (scalar) conductivities.

There are also available two choices of visualization types for the derived quantities. The node values can be calculated by taking the average of the derived values on neighbouring elements (constant weights). This results often in visually good images. The other possible choice is to weight the average with the size of the elements, which is more accurate and should be used when some other variable depends on these derived values. The latter choice is also the default.

16.4 Keywords

Solver solver id

Equation String Stat Current Solver

Variable String Potential

This may be of any name as far as it is used consistently also elsewhere.

Variable DOFs Integer 1

Degrees of freedom for the potential.

Procedure File "StatCurrentSolve" "StatCurrentSolver"

Following are listed two keywords with default values for output control.

Calculate Volume Current Logical [True]

Calculate Joule Heating Logical [True]

Constant Weights Logical [True]

Used to turn constant weighting on for the results.

Calculate Nodal Heating Logical [True]

Calculate nodal heating that may be used to couple the heat equation optimally when using conforming finite element meshes.

Power Control Real

Apply power control with the desired heating power being P_0 .

Current Control Real

Apply current control with the desired current being I_0 .

Material mat id

Electric Conductivity Real

Body Force bodyforce id

Current Source Real

Possibility for a current source, not used often though.

Joule Heat Logical

If this flag is active the Heat equation will automatically compute the quantity $\nabla\phi \cdot \sigma \nabla\phi$ as heat source. Then it is assumed that ϕ is named Potential. If there is no heat equation this flag has no effect.

Boundary Condition bc id

Potential Real

Dirichlet BC for the potential.

Current Density BC Logical

Must be set to True if Neumann BC is used.

Current Density Real

Neumann boundary condition for the current.

Model 17

Computation of Magnetic Fields in 3D

Module name: MagnetoDynamics

Module subroutines: WhitneyAVSolver, WhitneyAVHarmonicSolver, MagnetoDynamicsCalcFields

Module authors: Juha Ruokolainen, Mika Malinen, Eelis Takala

Document authors: Mika Malinen, Juha Ruokolainen, Juhani Kataja, Eelis Takala, Peter Råback

Document created: 9.9.2010

Document edited: 23.5.2019

17.1 Introduction

This module may be used to solve a version of the Maxwell equations in the \mathbf{A} - V form. The approximation of the associated vector potential variable \mathbf{A} is here done by using vector-valued (edge) finite element basis functions, while the classic Lagrange interpolation is applied to compute the scalar potential V . The use of edge finite elements limits the applicability of this solver to 3D problems. In addition to performing the computations in the time domain, the analogous version of the equations may also be solved in the frequency domain. Several ways to handle stationary equations are also included. Furthermore, an additional solver may be called to produce nodal approximations of derived fields after the two potentials have been obtained.

The equations over moving media are treated in the case of rigid motion. This implies that the associated velocity field is divergence-free and the velocity gradient is both constant and skew.

17.2 The transient equations

If a charge q moves in a region in which there are both electric and magnetic fields, the electromagnetic force on the charge moving with velocity \mathbf{v} is experimentally observed to be given by the Lorentz force

$$\mathbf{F} = q(\mathbf{E} + \mathbf{v} \times \mathbf{B}). \quad (17.1)$$

Alternatively, the force exerted on a volume element $d\Omega$ is therefore given by

$$d\mathbf{F} = (\rho\mathbf{E} + \rho\mathbf{v} \times \mathbf{B})d\Omega \quad (17.2)$$

where ρ is the charge density, per unit volume. By assuming a short relaxation time of the charge's motion as compared with the characteristic time scale of the motion of the body, we may incorporate the notion of such electromagnetic field in Ohm's law by writing it as

$$\mathbf{J} = \sigma\left(\frac{\mathbf{F}}{q}\right) \equiv \sigma\mathbf{E}', \quad (17.3)$$

where \mathbf{J} is the current density and σ is the electrical conductivity. An alternative would be to define the electromagnetic field \mathbf{E}' as the ratio of the volume force density to the charge density, but both the definitions

lead to the conclusion that in order to explain the Lorentz force we must have

$$\mathbf{E}' = \mathbf{E} + \mathbf{v} \times \mathbf{B}. \quad (17.4)$$

We note that since our definition of the electromagnetic field depends on the notions of force and charge which are assumed to be invariant quantities (under a change of observer), the electromagnetic field also has an invariant nature. It is tacit here that if the separation of the known electromagnetic field into its electric and magnetic components was sought via (17.4), the result would depend on the observer. However, we shall not need such separation since solving for \mathbf{E}' is enough to find the current density.

Otherwise the content of electromagnetic equations on a region Ω is here cast into a magnetic version of Maxwell's equations

$$\mathbf{curl} \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t}, \quad (17.5)$$

$$\mathbf{curl}\left(\frac{1}{\mu}\mathbf{B}\right) - \mathbf{J} = \mathbf{g}, \quad (17.6)$$

$$\mathbf{div} \mathbf{B} = 0, \quad (17.7)$$

$$\mathbf{div} \mathbf{J} = 0, \quad (17.8)$$

where μ is the permeability and \mathbf{g} is a divergence-free source satisfying (for ways to ensure this condition beforehand, see the description of the Helmholtz projection in the context of stationary equations below)

$$\mathbf{div} \mathbf{g} = 0. \quad (17.9)$$

Writing this system in terms of the electromagnetic field defined by (17.4) via eliminating \mathbf{E} and using Ohm's law (17.3) yields

$$\frac{\partial \mathbf{B}}{\partial t} - \mathbf{curl}(\mathbf{v} \times \mathbf{B}) + \mathbf{curl} \mathbf{E}' = \mathbf{0}, \quad (17.10)$$

$$\mathbf{curl}\left(\frac{1}{\mu}\mathbf{B}\right) - \sigma \mathbf{E}' = \mathbf{g}, \quad (17.11)$$

$$\mathbf{div} \mathbf{B} = 0, \quad (17.12)$$

$$\mathbf{div}(\sigma \mathbf{E}') = 0. \quad (17.13)$$

The content of this system is adequate for finding the instantaneous spatial solutions $\mathbf{B}(\mathbf{x}, t)$ and $\mathbf{E}'(\mathbf{x}, t)$ at time t simultaneously over both static and moving parts of the model. Here the point \mathbf{x} is thus specified by its coordinates with respect to a single, fixed Cartesian coordinate frame.

If an Elmer model contains moving bodies, the spatial discretization is typically done over an instantaneous configuration whose representation is obtained via a rigid motion of the previous mesh into the current configuration. In this case the time stepping machinery of Elmer by default produces an approximation to the material time derivative of a spatial solution (the material point is held fixed in the differentiation). It agrees with the usual definition of the substantial (or total) time derivative of the spatial field given as

$$\frac{D\mathbf{B}}{Dt} \equiv \frac{\partial \mathbf{B}}{\partial t} + (\mathbf{v} \cdot \nabla)\mathbf{B}. \quad (17.14)$$

Since the computational procedure does not make it straightforward to evaluate the standard partial derivatives with respect to time, the need of rewriting the equations in forms which employ other time derivatives arises as a natural endeavour. To this end, we shall consider the upper convected time derivative

$$\frac{\mathcal{D}\mathbf{B}}{\mathcal{D}t} = \frac{\partial \mathbf{B}}{\partial t} + (\mathbf{v} \cdot \nabla)\mathbf{B} - (\mathbf{B} \cdot \nabla)\mathbf{v} + (\mathbf{div} \mathbf{v})\mathbf{B}, \quad (17.15)$$

which agrees with the Lie derivative $L_{\mathbf{v}}\mathbf{B}$ of \mathbf{B} with respect to \mathbf{v} when the motion is rigid so that

$$\mathbf{div} \mathbf{v} = 0. \quad (17.16)$$

It can be expressed in terms of the substantial time derivative as

$$\frac{\mathcal{D}\mathbf{B}}{\mathcal{D}t} = \frac{D\mathbf{B}}{Dt} - (\mathbf{B} \cdot \nabla)\mathbf{v} + (\operatorname{div} \mathbf{v})\mathbf{B}. \quad (17.17)$$

By using vector identities, the definition (17.15) can also be rewritten as

$$\frac{\mathcal{D}\mathbf{B}}{\mathcal{D}t} = \frac{\partial \mathbf{B}}{\partial t} - \operatorname{curl}(\mathbf{v} \times \mathbf{B}) + \mathbf{v}(\operatorname{div} \mathbf{B}). \quad (17.18)$$

This immediately implies that the content of the Faraday law (17.10) can be expressed under Gauss's law for the magnetic field (17.12) as

$$\frac{\mathcal{D}\mathbf{B}}{\mathcal{D}t} + \operatorname{curl} \mathbf{E}' = \mathbf{0}. \quad (17.19)$$

Although we consider (17.19) to be a convenient way to express the Faraday law for our purposes, we do not want to assert whether this useful mathematical reduction happens by chance or whether it should be considered to reflect some theoretical underpinning.

In practice the solution is sought in terms of potentials. In the first place we set

$$\mathbf{B} = \operatorname{curl} \mathbf{A}. \quad (17.20)$$

It should be noted that this implies some ambiguity, since the resulting set of equations does not have a unique solution without imposing additional constraints on $\mathbf{A}(\mathbf{x}, t)$. Otherwise, if \mathbf{A} satisfies the equations, any field \mathbf{A}_ϕ having the decomposition $\mathbf{A}_\phi = \mathbf{A} + \nabla\phi$ may also be made to be a solution. The uniqueness of \mathbf{A} could be assured for example by seeking $\mathbf{A}(\cdot, t) \in H(\operatorname{curl}, \Omega) \cap H(\operatorname{div}, \Omega)$ that satisfies additionally

$$\operatorname{div} \mathbf{A} = 0$$

on Ω and

$$\mathbf{A} \cdot \mathbf{n} = 0$$

on the boundary $\partial\Omega$.

In the case of the vector potential description (17.20) the Faraday law in the form (17.10) yields

$$\operatorname{curl}\left(\frac{\partial \mathbf{A}}{\partial t} - \mathbf{v} \times \operatorname{curl} \mathbf{A} + \mathbf{E}'\right) = \mathbf{0},$$

so we may automate the satisfaction of the Faraday law by seeking the solution in the form

$$\frac{\partial \mathbf{A}}{\partial t} - \mathbf{v} \times \operatorname{curl} \mathbf{A} + \mathbf{E}' = -\nabla V, \quad (17.21)$$

with V being an unknown scalar potential. Switching to the substantial time derivative then gives

$$\frac{D\mathbf{A}}{Dt} - (\mathbf{v} \cdot \nabla)\mathbf{A} - \mathbf{v} \times \operatorname{curl} \mathbf{A} + \mathbf{E}' = -\nabla V. \quad (17.22)$$

Straightforward calculation shows that this is equivalent to

$$\frac{D\mathbf{A}}{Dt} - (\nabla \mathbf{A})^T \mathbf{v} + \mathbf{E}' = -\nabla V \quad (17.23)$$

and, by using the vector identity

$$(\nabla \mathbf{A})^T \mathbf{v} = \nabla(\mathbf{v} \cdot \mathbf{A}) - (\nabla \mathbf{v})^T \mathbf{A}, \quad (17.24)$$

we further obtain

$$\frac{D\mathbf{A}}{Dt} + (\nabla \mathbf{v})^T \mathbf{A} + \mathbf{E}' = -\nabla(V - \mathbf{v} \cdot \mathbf{A}). \quad (17.25)$$

In the case of rigid motion the velocity gradient is skew, so

$$(\nabla \mathbf{v})^T = -\nabla \mathbf{v} \quad (17.26)$$

and hence (17.25) finally gives

$$\frac{D\mathbf{A}}{Dt} - (\nabla\mathbf{v})\mathbf{A} + \mathbf{E}' = -\nabla V' \quad (17.27)$$

where we have set

$$V' = V - \mathbf{v} \cdot \mathbf{A}. \quad (17.28)$$

Interestingly, the content of (17.27) can also be expressed as

$$\frac{\mathcal{D}\mathbf{A}}{\mathcal{D}t} + \mathbf{E}' = -\nabla V'. \quad (17.29)$$

The equation (17.27) offers the possibility of eliminating the field \mathbf{E}' from the set of primary unknowns while the additional constraint (17.13) serves the determination of V' . It should be noted that since there is inherent ambiguity in the definition of the vector potential, the redefinition (17.28) of the scalar potential may not be of great importance in practice. Therefore we can just seek a solution $V'(\cdot, t) \in \mathcal{V} \subset H^1(\Omega)$.

To derive a weak version of the equations, let v be an appropriate test function for V' , so that we have $v \in H^1(\Omega)$. Multiplying (17.13) with v , integrating by parts and using (17.27) bring us to the weak formulation

$$\int_{\Omega} \sigma \left[\frac{D\mathbf{A}}{Dt} - (\nabla\mathbf{v})\mathbf{A} \right] \cdot \nabla v \, d\Omega + \int_{\Omega} \sigma \nabla V' \cdot \nabla v \, d\Omega = - \int_{\partial\Omega} (\sigma \mathbf{E}') \cdot \mathbf{n} v \, dS. \quad (17.30)$$

The determination of the scalar potential V' is thus joined with the possibility of specifying either V' or the normal component of the current density $\sigma \mathbf{E}'$ on the boundary. If the normal component of the current density is specified on the entire boundary $\partial\Omega$ as

$$-(\sigma \mathbf{E}') \cdot \mathbf{n} = j_n, \quad (17.31)$$

the boundary data must satisfy the compatibility condition

$$\int_{\partial\Omega} j_n \, dS = 0.$$

On the other hand, by using (17.20) and (17.27), we may rewrite (17.11) as

$$\sigma \left[\frac{D\mathbf{A}}{Dt} - (\nabla\mathbf{v})\mathbf{A} \right] + \sigma \nabla V' + \mathbf{curl} \left(\frac{1}{\mu} \mathbf{curl} \mathbf{A} \right) = \mathbf{g} \quad (17.32)$$

to obtain the weak version

$$\begin{aligned} \int_{\Omega} \sigma \left[\frac{D\mathbf{A}}{Dt} - (\nabla\mathbf{v})\mathbf{A} \right] \cdot \boldsymbol{\eta} \, d\Omega + \int_{\Omega} \sigma \nabla V' \cdot \boldsymbol{\eta} \, d\Omega + \int_{\Omega} \frac{1}{\mu} \mathbf{curl} \mathbf{A} \cdot \mathbf{curl} \boldsymbol{\eta} \, d\Omega \\ + \int_{\partial\Omega} \left(\frac{1}{\mu} \mathbf{curl} \mathbf{A} \right) \cdot (\boldsymbol{\eta} \times \mathbf{n}) \, dS = \int_{\Omega} \mathbf{g} \cdot \boldsymbol{\eta} \, d\Omega, \end{aligned} \quad (17.33)$$

with $\boldsymbol{\eta}$ an appropriate test function corresponding to \mathbf{A} . The weak formulations obtained from (17.33) and (17.30) generally form the basis for the A-V formulation of the problem.

It should be noted that here we have avoided seeking a unique solution in the space

$$\mathbf{A}(\cdot, t) \in H(\mathbf{curl}, \Omega) \cap H(\text{div}, \Omega)$$

as only the requirement $\mathbf{A}(\cdot, t) \in H(\mathbf{curl}, \Omega)$ appears to be necessary in this derivation. The Elmer implementation relies on this minimal regularity assumption so that a finite element approximation $\mathbf{A}_h(\cdot, t)$ is sought from an edge finite element space $X_h \subset H(\mathbf{curl}, \Omega)$. This simplification however leads to the inconvenience that the uniqueness of the vector potential solution cannot be ensured. Thus, given a solution (\mathbf{A}, V') , we can generate another solution $(\mathbf{A}_\phi, V_\phi) \equiv (\mathbf{A} + \nabla\phi, V' - \partial_t\phi)$, although the associated fields \mathbf{B} and \mathbf{E}' remain invariant.

The steady state solution of the evolutionary equations. Sometimes the steady state solution of the evolutionary model may be of an interest. For example the steady state solution of a case where a homogeneous disk is spun with a constant angular velocity in a region in which there is a steady magnetic field without circumferential variation can be described over a fixed domain. Then (17.21) yields

$$\mathbf{E}' - \mathbf{v} \times \mathbf{curl} \mathbf{A} = -\nabla V. \quad (17.34)$$

When a body makes a rigid rotation about the line through the origin \mathbf{o} spanned by the angular velocity $\boldsymbol{\omega}$, the velocity field is described by the formula

$$\mathbf{v}(\mathbf{x}, t) = \mathbf{v}(\mathbf{o}, t) + \boldsymbol{\omega}(t) \times (\mathbf{x} - \mathbf{o}). \quad (17.35)$$

We also note that the term $(\nabla \mathbf{v})\mathbf{A}$ occurring in (17.27) has the following representation in terms of $\boldsymbol{\omega}$

$$(\nabla \mathbf{v})\mathbf{A} = \boldsymbol{\omega} \times \mathbf{A}. \quad (17.36)$$

Solution in the frequency domain. The equations in the A-V form may also be solved in the frequency domain by assuming $\mathbf{v} = \mathbf{0}$. In this case the ansatz $\mathbf{A}(x, t) = \mathbf{A}(x) \exp(i\omega t)$ and $V(x, t) = V(x) \exp(i\omega t)$ is made to obtain the analogous set of equations for determining the complex-valued amplitudes $\mathbf{A}(x)$ and $V(x)$.

17.3 The stationary equations and the Helmholtz projection of a source

In the stationary case the electric field \mathbf{E}' is simply the gradient of a scalar potential. The weak formulation based on (17.33) and (17.30) then simplifies to

$$\int_{\Omega} \sigma \nabla V \cdot \boldsymbol{\eta} \, d\Omega + \int_{\Omega} \frac{1}{\mu} \mathbf{curl} \mathbf{A} \cdot \mathbf{curl} \boldsymbol{\eta} \, d\Omega + \int_{\partial\Omega} \left(\frac{1}{\mu} \mathbf{curl} \mathbf{A} \right) \cdot (\boldsymbol{\eta} \times \mathbf{n}) \, dS = \int_{\Omega} \mathbf{g} \cdot \boldsymbol{\eta} \, d\Omega \quad (17.37)$$

and

$$\int_{\Omega} \sigma \nabla V \cdot \nabla v \, d\Omega = - \int_{\partial\Omega} (\sigma \mathbf{E}') \cdot \mathbf{n} \, v \, dS. \quad (17.38)$$

As the solution of V and \mathbf{A} can then be done sequentially by first solving for V , the basic scenario for applying the solver to stationary cases is that only the field \mathbf{A} is solved by employing the weak formulation

$$\int_{\Omega} \frac{1}{\mu} \mathbf{curl} \mathbf{A} \cdot \mathbf{curl} \boldsymbol{\eta} \, d\Omega + \int_{\partial\Omega} \left(\frac{1}{\mu} \mathbf{curl} \mathbf{A} \right) \cdot (\boldsymbol{\eta} \times \mathbf{n}) \, dS = \int_{\Omega} \mathbf{g}_S \cdot \boldsymbol{\eta} \, d\Omega \quad (17.39)$$

where \mathbf{g}_S denotes the static source.

It should be noted that the source should generally be divergence-free. The divergence-freeness of \mathbf{g}_S may be assured by setting $\mathbf{g}_S = \mathcal{P}(\mathbf{g}_U)$ where \mathbf{g}_U is the user-supplied source term and the Helmholtz projection $\mathcal{P}(\mathbf{g}_U) = \mathbf{g}_U - \nabla Q$, with $Q \in M \subset H^1(\Omega)$, is defined via the requirement

$$\int_{\Omega} \nabla \cdot (\mathbf{g}_U - \nabla Q) q \, d\Omega = 0 \quad (17.40)$$

for any admissible variation q of Q . Integration by parts is used to obtain the version

$$\int_{\Omega} \nabla Q \cdot \nabla q \, d\Omega = \int_{\Omega} \mathbf{g}_U \cdot \nabla q \, d\Omega - \int_{\partial\Omega} \mathcal{P}(\mathbf{g}_U) \cdot \mathbf{n} q \, dS. \quad (17.41)$$

If \mathbf{g}_U has already been obtained from a scalar field V^s as

$$\mathbf{g}_U = -\sigma \nabla V^s, \quad (17.42)$$

to obtain a close resemblance of \mathbf{g}_U and $\mathcal{P}(\mathbf{g}_U)$ (especially, $\mathcal{P}(\mathbf{g}_U) = \mathbf{g}_U$ when \mathbf{g}_U is already divergence-free) it is natural to set

$$\mathcal{P}(\mathbf{g}_U) \cdot \mathbf{n} = \mathbf{g}_U \cdot \mathbf{n} \quad (17.43)$$

on the boundary where the current density was prescribed. On the remaining part of the boundary where the Dirichlet constraint was specified for V^s the homogeneous Dirichlet constraint for the field Q may be used. However, by default a different strategy is applied by setting $Q = 0$ on parts where $|\mathbf{g}_U \cdot \mathbf{n}| \neq 0$. Then the boundary integral in (17.41) vanishes. The Helmholtz projection of the source may also be performed when the evolutionary version of the equations is handled.

Other types of source vectors may also be considered as the user may generally specify the source in the form

$$\mathbf{g} = \mathcal{P}(\mathbf{g}_U) + \mathbf{curl} \mathbf{M}^s - \sigma \nabla V^s \quad (17.44)$$

or, if the Helmholtz projection is not applied,

$$\mathbf{g} = \mathbf{g}_U + \mathbf{curl} \mathbf{M}^s - \sigma \nabla V^s. \quad (17.45)$$

Here \mathbf{M}^s is referred to as the magnetization. The last terms in (17.44) and (17.45) enable the direct generation of the source electric current density in terms of the source potential V^s without first computing \mathbf{g}_U from (17.42).

To conclude, we note that it is also possible to solve the stationary equations such that both \mathbf{A} and V are handled as unknowns and solved simultaneously by employing the variational equations

$$\int_{\Omega} \sigma \nabla V \cdot \boldsymbol{\eta} \, d\Omega + \int_{\Omega} \frac{1}{\mu} \mathbf{curl} \mathbf{A} \cdot \mathbf{curl} \boldsymbol{\eta} \, d\Omega + \int_{\partial\Omega} \left(\frac{1}{\mu} \mathbf{curl} \mathbf{A} \right) \cdot (\boldsymbol{\eta} \times \mathbf{n}) \, dS = \int_{\Omega} \mathcal{P}(\mathbf{g}_U) \cdot \boldsymbol{\eta} \, d\Omega \quad (17.46)$$

and

$$\int_{\Omega} \sigma \nabla V \cdot \nabla v \, d\Omega = - \int_{\partial\Omega} (\sigma \mathbf{E}') \cdot \mathbf{n} \, v \, dS. \quad (17.47)$$

17.4 The boundary conditions

We see from the weak versions (17.33) and (17.30) that, if the Dirichlet type constraints

$$\mathbf{A} \times \mathbf{n} = \mathbf{a} \times \mathbf{n} \quad \text{and} \quad V' = \hat{v} \quad (17.48)$$

are not given, we may specify the tangential components of the magnetic field $\mathbf{H} = (1/\mu)\mathbf{B}$ and the normal component of current density $\mathbf{J} = \sigma \mathbf{E}'$. These may be defined via giving \mathbf{h} to define

$$\frac{1}{\mu} \mathbf{curl} \mathbf{A} = \mathbf{n} \times \mathbf{h} \times \mathbf{n} \quad (17.49)$$

and

$$-(\sigma \mathbf{E}') \cdot \mathbf{n} = j_n. \quad (17.50)$$

We note that giving a Dirichlet constraint is a useful way to guarantee the uniqueness of the scalar potential V' which would otherwise be determined only up to a constant.

A Robin-like generalization of (17.49) leads to the boundary condition

$$\frac{1}{\mu} \mathbf{curl} \mathbf{A} = \alpha(\mathbf{A} \times \mathbf{n}) + \mathbf{n} \times \mathbf{h} \times \mathbf{n}, \quad (17.51)$$

with α being a given parameter. We however note that the case $\alpha \neq 0$ may lack in having a physical interpretation. On the other hand, a generalized version of (17.50) is written as

$$-(\sigma \mathbf{E}') \cdot \mathbf{n} = j_n - \beta V', \quad (17.52)$$

which in the stationary case reduces to the Robin boundary condition

$$\sigma \nabla V \cdot \mathbf{n} + \beta V = j_n. \quad (17.53)$$

In addition, a procedural technique may be applied to specify a Dirichlet constraint for \mathbf{A} when the normal component of the magnetic flux density \mathbf{B} is given on the boundary.

17.5 A consistently stabilized Lagrange multiplier formulation for magnetostatics

The possibility to seek a unique vector potential solution to the magnetostatic equations is included here as a special case via using the scalar variable to impose the divergence-free constraint on \mathbf{A} in a weak manner. This strategy employs the weak formulation

$$\begin{aligned} \int_{\Omega} \frac{1}{\mu} \operatorname{curl} \mathbf{A} \cdot \operatorname{curl} \boldsymbol{\eta} \, d\Omega + \int_{\Omega} \nabla \phi \cdot \boldsymbol{\eta} \, d\Omega &= \int_{\Omega} \mathcal{P}(\mathbf{g}_U) \cdot \boldsymbol{\eta} \, d\Omega, \\ \int_{\Omega} (\mathbf{A} - k \nabla \phi) \cdot \nabla q \, d\Omega &= 0 \end{aligned} \quad (17.54)$$

with $\phi \in L_2(\Omega)/P_0(\Omega)$ (the definition of the space $L_2(\Omega)/P_0(\Omega)$ reflects the fact that ϕ can be determined up to a freely chosen constant only) and k being a stabilization parameter. The choice $k = 1$ is seen to be natural. Then, if $\Pi \mathbf{A}$ denotes the Helmholtz projection of \mathbf{A} , the content of this formulation can also be cast into the form

$$\begin{aligned} \int_{\Omega} \frac{1}{\mu} \operatorname{curl} \mathbf{A} \cdot \operatorname{curl} \boldsymbol{\eta} \, d\Omega + \int_{\Omega} (\mathbf{A} - \Pi \mathbf{A}) \cdot (\boldsymbol{\eta} - \Pi \boldsymbol{\eta}) \, d\Omega &= \int_{\Omega} \mathcal{P}(\mathbf{g}_U) \cdot \boldsymbol{\eta} \, d\Omega, \\ \int_{\Omega} \Pi \mathbf{A} \cdot \nabla q \, d\Omega &= 0. \end{aligned} \quad (17.55)$$

This formulation reflects how the scheme works; the second term in the left-hand side of (17.55) acts as a penalization term to get into the right set of divergence-free solutions. That is, $\mathbf{A} - \Pi \mathbf{A} = \mathbf{0}$ is enforced by having this term.

17.6 Keywords

Keywords for WhitneyAVSolver

Here we list the keywords that are relevant to solving the evolutionary and stationary versions of the equations by calling the solver subroutine `WhitneyAVSolver` and that may also be common to the other solvers. Such common keywords relate to specifying material parameters, body forces, and boundary conditions.

Constants

`Permeability of Vacuum` `Real`

This constant has the default value $4\pi \cdot 10^{-7}$.

`Material` `mat id`

The following material parameters may be used by all the solvers in the module.

`Electric Conductivity` `Real`

This keyword is used to specify the electric conductivity σ . If the material is anisotropic, the electric conductivity may also be defined to be a tensor function by giving its components with respect to the global coordinate lines.

`Relative Permeability` `Real`

If this keyword is used, the permeability μ can be specified in terms of the permeability of vacuum. To obtain the permeability, the value of this keyword is then internally multiplied with the permeability of vacuum. Instead of using this keyword, the keywords `Permeability` or `Reluctivity` may be used.

Permeability Real

This keyword may be used to specify directly the permeability μ .

Reluctivity Real

The value of this keyword specifies the reluctivity ν . The permeability is then taken to be $\mu = 1/\nu$.

Solver solver id

Equation String WhitneySolver

A describing name for the discrete model handled by this solver may be given by using this keyword. The name can be changed as long as it is used consistently.

Procedure File "MagnetoDynamics" "WhitneyAVSolver"

This declaration specifies the name of the solver subroutine.

Variable String AV

The name of the variable may be freely chosen provided it is used consistently also elsewhere. The associated number of degrees of freedom should always be one.

Element

The default initialization routine should give a suitable element type definition automatically, so that the value of this keyword need not necessarily be given by the user.

Use Piola Transform Logical

Several types of edge finite elements can be used for spatial discretization. This keyword must be given the value `True` when the basis functions for the edge element interpolation are selected to be members of the optimal edge element family or when second-order approximation is used. The elements of the optimal family provide optimal accuracy even when the physical elements are not obtained as affine images of the reference elements. The simpler basis functions which are used otherwise may not provide such accuracy for non-affine element shapes. For the documentation of the edge element basis functions see the appendices of the ElmerSolver Manual.

Quadratic Approximation Logical

This keyword can be used to activate the approximation with the edge finite elements of second degree; see also the keyword `Use Piola Transform`.

Fix Input Current Density Logical

To ensure the divergence-freeness of the source term via performing the projection (17.40), the value `True` should be given for this keyword.

Automated Source Projection BCs Logical

If the projection (17.40) is applied to the user-specified source \mathbf{g}_U , the solver has originally constrained the field Q automatically such that Q is chosen to satisfy the zero Dirichlet condition on parts where $|\mathbf{g}_U \cdot \mathbf{n}| \neq 0$. This feature can be disabled by giving the value `False` for this keyword in order to specify, for example, the homogeneous Dirichlet constraint for Q (known as the field `Jfix`) on boundaries where the scalar potential V or V^s is constrained by a Dirichlet constraint.

Use Lagrange Gauge Logical

This keyword must be `True` in order to switch to a Lagrange multiplier formulation which has a unique vector potential solution.

Lagrange Gauge Penalization Coefficient Real

This keyword defines the stabilization parameter k in (17.54).

Use Tree Gauge Logical

Due to the chosen discretization, the vector potential is not sufficiently constrained to guarantee the uniqueness. Despite this, the iterative solvers are expected to be able to generate a consistent solution among many. However, to enable the solution with direct solvers this keyword is by default given the value `True` so that a special technique is applied to additionally constrain the discrete vector potential variable (in the case of iterative solvers the default value is `False`). This option is not supported when `Use Piola Transform = True` is given.

Linear System Refactorize Logical

It is noted that if the refactorization of the system matrix is controlled with this keyword, the matrix factorization is anyhow recomputed automatically if the time step size differs from the previous one.

Linear System Preconditioning String

Here the value `None` may give better performance than the standard ILU preconditioners since the null space of the system matrix may be non-trivial, leading to a singularity problem in connection with handling the LU decomposition.

Linear System Iterative Method String

The iterative solvers BiCGStab or BiCGStab(L) may work well.

Body Force bf id

In the body force section the user may give various volumetric sources contained in the vector \mathbf{g} as defined in either (17.44) or (17.45). The velocity field related to a rigid motion may also be defined.

Current Density i Real

This keyword is used to specify the components of the source \mathbf{g}_U .

Magnetization i Real

This keyword is used to specify the components of the magnetization M_i^s , with $i = 1, 2, 3$.

Electric Potential Real

This keyword specifies the source electric potential V^s .

Angular Velocity i Real

When a rigid motion is modelled, this keyword defines the components of the angular velocity $\boldsymbol{\omega}$. This gives also the representation for the term $(\nabla \mathbf{v})\mathbf{A}$; cf. the equation (17.36).

Lorentz Velocity i Real

This keyword may be used to define the velocity $\mathbf{v}(\mathbf{x}, t)$ when the steady state solution of the evolutionary model is sought. Alternatively, if the velocity is specified by the formula (17.35) and the angular velocity $\boldsymbol{\omega}$ is given, a constant value may be given in order to specify the uniform part $\mathbf{v}(\mathbf{o}, t)$ of the velocity.

Boundary Condition bc id

As explained, two versions of Dirichlet conditions are possible in connection with the A-V formulation. The first option relates to giving the value of the scalar potential, while the other version is used for prescribing the tangential components of the vector potential field. Assuming that the solver variable is AV, we may thus use the following keywords to specify the Dirichlet conditions:

AV Real

This keyword is used to specify the Dirichlet condition for the scalar potential V' , which is approximated by using the standard Lagrange interpolation.

AV {e} j Real

This keyword is used to give the vector \mathbf{a} in (17.48) in order to prescribe the degrees of freedom corresponding to the edge element interpolation of the vector potential. The value of this keyword defines the component a_j , $j \in \{1, 2, 3\}$, with respect to the global Cartesian coordinate system.

Jfix Real

This keyword is used to specify the Dirichlet condition for the scalar potential field Q , which is defined via (17.40).

The following keywords may be used in order to handle the flux-related boundary conditions:

Magnetic Field Strength i Real

This keyword can be used to define the components h_i of the vector \mathbf{h} so that the boundary conditions (17.49) and (17.51) may be imposed.

Electric Current Density Real

This keyword can be used to define the electric current density j_n in the boundary conditions (17.50) and (17.52).

Current Density Real

If the Helmholtz projection of the source is applied, this keyword may be used to specify the left-hand side in (17.43) as $\mathbf{g}_U \cdot \mathbf{n} = -j_n$, with j_n the value of this keyword.

Magnetic Transfer Coefficient Real

The value of this keyword gives the parameter α in the boundary condition (17.51).

Electric Transfer Coefficient Real

The value of this keyword gives the parameter β in the boundary condition (17.52).

Finally, the following keywords relate to a procedural technique to determine a tangential constraint for the vector potential \mathbf{A} when the normal component of the magnetic flux density \mathbf{B} is specified on the boundary.

Magnetic Flux Density i Real

This keyword is used to specify the components of the magnetic flux density \mathbf{B} with respect to the global Cartesian coordinate axes.

Magnetic Flux Density {n} Real

This keyword may be used to specify directly the normal component B_n of the magnetic flux density.

Keywords for WhitneyAVHarmonicSolver

In the following the additional keywords related to solving the harmonic version are listed. Typically these are used for giving optional values which specify the imaginary parts of the parameter values. The corresponding real parts are then given by using the keyword commands already described above.

Solver solver id

Equation String WhitneyHarmonicSolver

This gives a describing name for the discrete model handled here. The name can be changed as long as it is used consistently.

Procedure File "Magnetodynamics" "WhitneyAVHarmonicSolver"

The name of the solver subroutine is declared.

Variable String P[Pot re:1 Pot im:1]

The name of the variable may be freely chosen provided it is used consistently also elsewhere. The associated number of degrees of freedom is always two. Here the real and imaginary parts are named so that they are easily recognized.

Angular Frequency Real

The angular frequency $\omega = 2\pi f$ in the harmonic ansatz is specified.

Material mat id

Reluctivity Im Real

The reluctivity $\nu = 1/\mu$ may be specified to be a complex-valued quantity with the imaginary part given by using this keyword.

Electric Conductivity Im Real

The value of this keyword may be used to specify the imaginary part of the conductivity parameter. If the material is anisotropic, the electric conductivity may also be defined to be a tensor function by giving its components with respect to the global coordinate lines.

Body Force bf id

The following keywords are used to specify the imaginary parts of the volume sources:

Current Density Im i Real

Magnetization Im i Real

Boundary Condition bc id

The following keywords relate to specifying imaginary parts in conjunction with defining boundary conditions:

Magnetic Field Strength Im i Real

Electric Current Density Im Real

Magnetic Transfer Coefficient Im Real

Electric Transfer Coefficient Im Real

Magnetic Flux Density Im i Real

Magnetic Flux Density Im {n} Real

Keywords for MagnetoDynamicsCalcFields

An additional solver may finally be called to compute derived fields.

Solver solver id

The fields to be computed are chosen in the solver section. The field Magnetic Flux Density is computed always, others if requested. The size of a vector field is 3, while a tensor field has the size 6. For the harmonic solution the sizes are doubled as the imaginary components are also present.

Equation String CalcFields

A describing name for the solver is given. This can be changed as long as it is used consistently.

Procedure File "MagnetoDynamics" "MagnetoDynamicsCalcFields"

The name of the solver subroutine is given.

Potential Variable String

This keyword is used to specify the name of the underlying potential variable, for example AV. For edge elements the Use Piola Transform and Quadratic Approximation flags are herited from the solver owning this variable.

Angular Frequency Real

The angular frequency must be declared in this connection also as this is needed by some post-processed fields.

Calculate Magnetic Field Strength Logical

If True is given, a vector field Magnetic Field Strength is computed.

Calculate Electric Field Logical

If True is given, a vector field Electric Field is computed.

Calculate Current Density Logical

If True is given, a vector field Current Density is computed.

Impose Body Force Current Logical

If True is given, the body force current density given by the keyword Current Density i is added to above.

Calculate Maxwell Stress Logical

If True is given, a tensor field Maxwell Stress is computed.

Calculate Harmonic Loss Logical

If True is given, scalar fields Harmonic Loss Linear and Harmonic Loss Quadratic are computed. See Chapter 24 for more details.

Calculate Joule Heating Logical

If True is given, a scalar field Joule Heating is computed.

Calculate Nodal Heating Logical

If True is given, a scalar field Nodal Joule Heating is computed. In SI system the resulting unit is Watt. The nodal heating is easy to directly link as a nodal heat source to the heat equation assuming conforming meshes for the both equations.

Calculate Nodal Fields Logical

If this is set False do not compute nodal fields at all. The default is True.

Calculate Elemental Fields Logical

If this is set False do not compute elemental fields at all. The default is True. Elemental fields are nice in that they can present discontinuities in meshes.

If the harmonic loss will be computed then the following material parameters should be provided. Note that these are not parameters needed by the primary solver since the loss estimation is just a post-processing feature. See Chapter 24 for more details.

Material material id

Harmonic Loss Linear Coefficient Real

This keyword is used to define the material parameter C in (24.6) for the losses that are linear with frequency. Note that the coefficient may be a function of frequency itself.

Harmonic Loss Quadratic Coefficient Real

As the previous keyword except define the material parameter C for the quadratic loss terms.

Component component id

Calculate Magnetic Force Logical

The lumped magnetic force affecting the master bodies of the current component will be calculated if this is set true. If elemental fields are available, then also airgap forces are also lumped.

Calculate Magnetic Torque Logical

The total torque associated with the master bodies of the current component will be calculated if this is set true. If elemental fields are available, then also torque arising from airgap forces is included.

Torque Origin(3) Real

This keyword is used to define a point in space which torque axis meets.

Torque Axis(3) Real

This keyword is used to define the axis with respect to which torque is calculated.

In addition to the field computation, two scalar quantities are always computed by the solver and saved in the list of the Simulation section values: Eddy current power and Magnetic Field Energy. The first one is only relevant for time-dependent and harmonic cases.

Model 18

Circuits and Dynamics Solver

Module name: CircuitsAndDynamics

Module subroutines: CircuitsAndDynamics, CircuitsAndDynamicsHarmonic, CircuitsOutput

Module authors: Eelis Takala, Juha Ruokolainen

Document authors: Eelis Takala

Document created: May 23rd 2016

18.1 Introduction

It may become necessary to connect magnetoquasistatic model to circuit equations when modeling electrical devices that are circuit driven. Even if the model could in theory be handled without introducing circuits, it can considerably simplify the model set up. In Elmer the circuit equations are added in the system matrix of magnetodynamics solver and thus they are strongly solved together with the finite element equations. One should note that by adding the circuit equations may hinder the parallel performance of the solver. In case this is a critical concern to the user, one should pay attention in tuning the model [9] (for example using the so called reduced support that can bring great benefits [8]). Here the module for adding the circuit equations to the system matrix of the magnetoquasistatic solvers is presented. Note that this solver may (and needs to) be used together with the modules `MagnetoDynamics` or `MagnetoDynamics2D`.

When the `CircuitsAndDynamics` module is used in published academic work please refer to [9] and when reduced support is used in order to achieve better parallel scalability one may refer to [8].

18.2 Theory

The magneto dynamic problems in Elmer are currently solved by using the so called $a - v$ formulation. This can be done in 2D or 3D and with all simulation types (steady/harmonic/transient). The $a - v$ formulation [7, p. 124] may be presented as follows: find $\vec{a} \in W_e^1$ and $v \in W_e^0$, such that

$$(\nu \nabla \times \vec{a}, \nabla \times \vec{a}')_{\Omega} + \partial_t(\sigma \vec{a}, \vec{a}')_{\Omega_c} + (\sigma \nabla v, \vec{a}')_{\Omega_c} + (\vec{j}_0, \vec{a}')_{\Omega} + \langle (\vec{n} \times \vec{h}), \vec{a}' \rangle_{\Gamma_h} = 0 \quad \forall \vec{a}' \in W_e^1 \quad (18.1)$$

and

$$\partial_t(\sigma \vec{a}, \nabla v')_{\Omega_c} + \partial_t(\sigma \nabla v, \nabla v')_{\Omega_c} + \langle \vec{n} \cdot \vec{j}, v' \rangle_{\Gamma_c} = 0 \quad \forall v' \in W_e^0, \quad (18.2)$$

where $\nu = \mu^{-1}$ is the reluctivity, \vec{a} is the magnetic vector potential, v is the electric scalar potential, \vec{j}_0 is the source current density, \vec{n} is the normal vector of the boundary Γ_h , \vec{a}' is a test function in W_e^1 and v' is a test function in W_e^0 . The domain Ω is the considered space, $\Omega_c \subset \Omega$ is the space of conductive material, Γ_h is the boundary where the magnetic field is imposed and Γ_c is the boundary where current density field is imposed. The edge element discretization [7, p. 93] generates W_e^1 and W_e^0 discrete function spaces of the magnetic vector potential \vec{a} and electric scalar potential v functions. This formulation requires a system matrix with two types of unknowns, namely the magnetic vector potential and the electric potential. For more information about the formulation in Elmer see the modules `MagnetoDynamics` or `MagnetoDynamics2D`.

18.2.1 Component and Circuit network equations

The circuit equations are described with a general equation

$$\mathbf{A}\vec{x}' + \mathbf{B}\vec{x} = \vec{f} \quad (18.3)$$

where \vec{x} is the circuit variable vector, \mathbf{A} and \mathbf{B} are the coefficient matrices and \vec{f} is the force vector. In Elmer this equation can be written to the solver input file (SIF) in MATC language.

Circuit equations can be divided into two categories: 1. component equations that determine the behavior of a component and 2. circuit network equations that determine the relationship between the components (the Kirchhoff I and II laws).

The component equations are further divided into two subcategories: 1. the circuit element equations that can be used to define ideal components (for example Ohm's law) and 2. FE component equations that are coupled to the finite element equations (of those elements which belong to the corresponding components). During the development of the CircuitsAndDynamics module a new keyword section, called the "Component" section, was added. Within this section the user may control the component equation by specifying which bodies belong to the component, which formulation should be used with it and inputting component formulation specific data (for example number of turns).

The circuit network equations are further divided into two categories as well: 1. Kirchhoff I law that is the current conservation law (every node in the network graph must conserve the current) and 2. Kirchhoff II law that is the conservation of energy (the potential difference over every loop in the network graph must be zero).

In Elmer the circuit equations are technically divided into two categories: 1. the automatically written equations and 2. the manually written equations. This is due to the fact that on one hand it is convenient to write the FE component equations automatically once the formulation and its specific data is known and on the second hand the rest of the equations can be manually written without any difficulties. Note that the ideal components could be automated as well (for example a resistor model), however such components do not exist at the moment and the user needs to write those equations manually.

In the following sections the three supported component formulations (massive, stranded, foil winding) are presented. The type of the formulation can be chosen in SIF via "Coil Type" keyword in the "Component" section.

18.2.2 Massive coil

There may be some cases, where the winding only consists of few turns, and thus could be computed using the so called massive inductor model [2] where $a - v$ formulation takes the form: for all j_m

$$(\nu \nabla \times \vec{a}, \nabla \times \vec{a}')_{\Omega} + \partial_t(\sigma \vec{a}, \vec{a}')_{\Omega_m} + \sum_{i \in \Gamma_{j_m}} V_{j_m}(\sigma \nabla v_0^i, \vec{a}')_{\Omega_m} = 0 \quad (18.4)$$

and

$$\partial_t(\sigma \vec{a}, \nabla s^i)_{\Omega_m} + V_{j_m}(\sigma \nabla v_0^i, \nabla s^i)_{\Omega_m} = I_{j_m}, \quad (18.5)$$

where j_m is the massive inductor index number (there may exist multiple instances in one computation), Ω_m is the massive inductor domain, Γ_{j_m} is the electrode boundary, V_{j_m} is the voltage and I_{j_m} is the total current through Γ_{j_m} .

The source electric potential v_0^i can be precomputed with electrokinetic formulation (Poisson equation) by setting it 1 at the positive boundary and 0 at the negative electrode boundary. If the electrokinetic solver is not executed then the reduced support is used.

18.2.3 Stranded coil

In case of a coil with a small diameter wire that is densely packed, a stranded model could be used. The main line of models takes the form [3]: for all j_s

$$(\nu \nabla \times \vec{a}, \nabla \times \vec{a}')_{\Omega} + I_{j_s}(\vec{j}_{s,j_s}, \vec{a}')_{\Omega_s} = 0 \quad (18.6)$$

and

$$\partial_t(\sigma \vec{a}, \vec{j}_{s,j_s})_{\Omega_s} + I_{j_s}(\sigma \vec{j}_{s,j_s}, \vec{j}_{s,j_s})_{\Omega_s} = V_{j_s}, \quad (18.7)$$

where j_s is the stranded inductor index number, I_{j_s} is the total current in the coil and V_{j_s} is the voltage in the coil. This is the preferred method for defining the component equation for the In Elmer the so called classical stranded inductor model [7, p. 286], [6, 5], where the resistance of the coil is explicitly given, can also be used. The classical model yields

$$\partial_t \int_{\Omega_s} \vec{a} \cdot \vec{w} \, d\Omega_s + R I_{j_s} = V_{j_s}, \quad (18.8)$$

where \vec{w} is the wire density vector and R is the resistance of the coil.

18.2.4 Foil winding model

In case of a foil winding with a small layer thickness that is densely packed, a foil winding model could be used. It takes the following form [4, 1]: for all j_f

$$(\nu \nabla \times \vec{a}, \nabla \times \vec{a}')_{\Omega} + \partial_t(\sigma \vec{a}, \vec{a}')_{\Omega_f} + \sum_{i \in \Gamma_{j_f}} (\sigma V_{j_f}(\alpha) \nabla v_0^i, \vec{a}')_{\Omega_f} = 0 \quad (18.9)$$

and

$$\partial_t(\sigma \vec{a}, V'(\alpha) \nabla s^i)_{\Omega_f} + (\sigma V_{j_f}(\alpha) \nabla v_0^i, V'(\alpha) \nabla v')_{\Omega_f} = \frac{N_{j_f}}{L_{j_f}} I_{j_f} \int_{\Omega_{\alpha,j_f}} V'(\alpha) \, d\alpha, \quad (18.10)$$

where j_f is the foil winding index number, Ω_f is the massive inductor domain, Γ_{j_f} is the electrode boundary, V_{j_f} is the voltage and I_{j_f} is the total current through Γ_{j_f} .

18.3 Keywords

The circuits and dynamics solver adds the circuit equations to system matrix of the WhitneySolver or the MagnetoDynamics2D solver. Keywords are needed to both of these solvers.

18.3.1 CircuitsAndDynamics solver

Solver `solver id`

Equation `String Circuits`

Procedure `File "CircuitsAndDynamics" "CircuitsAndDynamics"`

The procedure which includes the addition of circuit equations to the linear system of MagnetoDynamics and MagnetoDynamics2D solvers.

No Matrix `Logical True`

This solver does not have it's own matrix. It only operates on the matrices of the magneto dynamics solvers.

Component `component id`

Master Bodies `Integer body ids`

Body ids of the bodies which are defined as a circuit component.

Coil Type `String type`

If the circuit component is defined as a coil, it can be treated as a "stranded", "massive" or "foil winding".

Resistance `Real resistance`

If the coil type is defined stranded then the resistance of the coil can explicitly be defined with this keyword. In case this keyword is not given then the resistance is computed by using the electric conductivity of the associated body materials.

Electrode Area Real area

If the coil type is stranded and resistance is not explicitly given, then this keyword may be given to specify the area of the coil terminal. This is then used to compute the Resistance of the coil. Note that if neither the Resistance nor the Electrode Area are given then the area of the terminal is automatically computed. However, at the moment in 3D this feature is not yet working.

Body Force 1

Source Name Real value

By defining this the source "Source Name" can be used in the circuit definitions. The name of the source can of course be freely chosen.

18.3.2 CircuitsAndDynamics output

This solver saves the results of all the variables that belong to the CircuitsAndDynamics solver, namely currents and voltages of components (also those variables that are defined by the user). These can then be saved into a text file by using the SaveData solver.

Simulation

Max Output Level Integer Level

This determines what is shown on the standard output. Level 3 shows some information what is happening during the execution, Level 8 shows component variable results and Level 10 shows all the Circuit Variable results. All these variables are stored even if they are not shown in the standard output. The stored variables can be saved with the SaveData routine.

Solver solver id

Equation String Circuits Output

Procedure File "CircuitsAndDynamics" "CircuitsOutput"

The procedure which outputs the results of the added circuits (currents, voltages, resistances, etc...).

18.3.3 Additional Keywords to the MagnetoDynamics or MagnetoDynamics2D solvers

Solver solver id

Equation String WhitneySolver

Procedure File "MagnetoDynamics" "WhitneyAVSolver"

This is the WhitneySolver to whose system matrix the circuit equations are added. This procedure could be MagnetoDynamics2D as well.

Export Lagrange Multiplier Logical True

This is needed in the solvers when circuit equations are added.

18.3.4 Circuit Definitions

The circuit definitions are written in MATC.

```
$ nof_circuits = 1
$ nof_variables = 6
$ Circuits = nof_circuits

$ !----- Circuit 1 ----
$ ! Define variable count and initialize circuit matrices
$ C.1.variables = nof_variables
$ C.1.A = zeros(nof_variables, nof_variables)
```

```

$ C.1.B = zeros(nof_variables, nof_variables)
$ C.1.Mre = zeros(nof_variables, nof_variables)
$ C.1.Mim = zeros(nof_variables, nof_variables)
$ C.1.perm = zeros(nof_variables)

$ ! Define variables
$ C.1.name.1 = "var1"
$ C.1.name.2 = "var2"

$ ! Define component variables
$ C.1.name.3 = "i_component(1)"
$ C.1.name.4 = "v_component(1)"
$ C.1.name.5 = "i_component(2)"
$ C.1.name.6 = "v_component(2)"
! The number in the parenthesis refers to
! the component id that is defined in sif.

$ ! Define Sources:
$ C.1.B(0,0) = 1
$ C.1.source.1 = "Source Name"

```

Bibliography

- [1] Patrick Dular and Christophe Geuzaine. Spatially dependent global quantities associated with 2-d and 3-d magnetic vector potential formulations for foil winding modeling. *IEEE trans. magn.*, 38(2):633–636, March 2002.
- [2] Patrick Dular, F. Henrotte, and W. Legros. A general and natural method to define circuit relations associated with magnetic vector potential formulations. *IEEE trans. magn.*, 35(3):1630–1633, May 1999.
- [3] Patrick Dular, Nelson Sadowski, J.P.A. Bastos, and Willy Legros. Dual complete procedures to take stranded inductors into account in magnetic vector potential formulations. *IEEE trans. magn.*, 36(4):1600–1605, July 2000.
- [4] H. D. Gersem and K. Hameyer. A finite element model for foil winding simulation. *IEEE trans. magn.*, 37(5):3427–3432, September 2001.
- [5] C. Golovanov, Y. Marechal, and G. Meunier. 3d edge element based formulation coupled to electric circuits. *IEEE trans. magn.*, 34(5):3162–3165, September 1998.
- [6] P. J. Leonard and D. Rodger. Voltage forced coils for 3d finite element method electromagnetic model. *IEEE trans. magn.*, 24(5):2579–2581, September 1988.
- [7] Gerard Meunier, editor. *The Finite Element Method for Electromagnetic Modeling*. iSTE and Wiley, 2008.
- [8] Eelis Takala, Evren Yurtesen, Jan Westerholm, Juha Ruokolainen, and Tommi Peussa. Using reduced support to enhance parallel strong scalability in 3d finite element magnetic vector potential formulations with circuit equations. *Electromagnetics*, 36(6):400–408, August 2016.
- [9] Eelis Takala, Evren Yurtesen, Jan Westerholm, Juha Ruokolainen, and Peter Råback. Parallel simulations of inductive components with elmer finite element software in cluster environments. *Electromagnetics*, 36(3):167–185, April 2016.

Model 19

Vectorial Helmholtz for Electromagnetic Waves

Module name: VectorHelmholtz

Module subroutines: VectorHelmholtzSolver, VectorHelmholtzCalcFields

Module authors: Juhani Kataja, Juha Ruokolainen and Mika Malinen

Document authors: Juhani Kataja and Roman Szewczyk

Document created: Nov 6, 2014

Document edited: Apr 21, 2015

19.1 Introduction

This module is aimed to solve time-harmonic Maxwell's equations in high-frequency regime so that the curl-curl equation is valid.

19.2 Theory

The time-harmonic Maxwell's equations (time factor $e^{-i\omega t}$, $\omega = 2\pi f$) are given by

$$\mu^{-1} \nabla \times \vec{E} = i\omega \vec{H} \quad (19.1)$$

$$\nabla \times \vec{H} = -i\omega \varepsilon \vec{E} + \vec{J}, \quad (19.2)$$

where $\mu = \mu_0 \mu_r$ and $\varepsilon = \varepsilon_0 \varepsilon_r$ are the permeability and permittivity models of the system, both complex scalars, \vec{E} , \vec{H} and \vec{J} are electric field, magnetic field strength and impressend current distribution, respectively. The quantities μ_0, ε_0 are the permeability and permittivity of vacuum, and μ_r, ε_r are the relative permeability and permittivity.

The computational domain is denoted with Ω and $\partial\Omega = \overline{\Gamma_E} \cup \overline{\Gamma_Z}$.

Factoring out \vec{H} from the equations (19.1) and (19.2) results in

$$\nabla \times \mu^{-1} \nabla \times \vec{E} - \omega^2 \varepsilon \vec{E} = i\omega \vec{J}. \quad (19.3)$$

The Dirichlet and Robin boundary conditions for (19.3) are

$$\vec{n} \times \vec{E} = f \quad \text{on } \Gamma_E, \quad (19.4)$$

$$\vec{n} \times \nabla \times \vec{E} - \alpha \vec{n} \times (\vec{n} \times \vec{E}) = g \quad \text{on } \Gamma_Z, \quad (19.5)$$

respectively. Note that the Neumann boundary condition is achieved by omitting α or setting it 0.

The variational form of (19.3) is

$$\begin{aligned} \text{Find } \vec{E} \in \mathbf{H}_{f,E}(\nabla \times) \text{ so that} \\ \int_{\Omega} \mu^{-1} \nabla \times \vec{E} \cdot \nabla \times \vec{v} - \omega^2 \varepsilon \vec{E} \cdot \vec{v} d\Omega - \alpha \int_{\Gamma_Z} \mu^{-1} (\vec{n} \times \vec{E}) \cdot \vec{n} \times \vec{v} d\sigma \\ = - \int_{\Gamma_Z} \mu^{-1} g \cdot \vec{v} d\sigma + \int_{\Omega} i\omega \vec{J} \cdot \vec{v} d\Omega \quad \forall \vec{v} \in H_{0,E}(\nabla \times). \end{aligned} \quad (19.6)$$

19.2.1 Boundary condition models

Leontovich impedance boundary The Robin boundary condition (19.5) can be utilized to implement impedance boundary approximation of well conducting medium by choosing

$$\alpha = -i\omega\mu Z_p^{-1} \quad \text{and} \quad g = 0, \quad (19.7)$$

where Z_p is the surface impedance. Where Z_p can be given as, e.g.,

$$Z_p = (1 - i) \sqrt{\frac{\mu_c \omega}{2\sigma_c}}, \quad (19.8)$$

where σ_c and μ_c are the bulk conductivity and permeability of the wall [1].

First order absorbing boundary condition Let now the computational domain Ω be $B_R \setminus D$, where $B_R \subset \mathbb{R}^3$ is an open ball of radius R and $D \subset B_R$. Furthermore, suppose that $\Gamma_E \cup \Gamma_Z \subset \partial D$.

The first order absorbing boundary condition on ∂B_R is given by [2],

$$\vec{n} \times \nabla \times \vec{E} = i\omega \sqrt{\varepsilon_0 \mu_0} \vec{n} \times (\vec{n} \times \vec{E}), \quad \text{on } \partial B_R. \quad (19.9)$$

This can be recovered with a Robin boundary on ∂B_R with

$$\alpha = i\omega \sqrt{\varepsilon_0 \mu_0} \quad \text{and} \quad g = 0. \quad (19.10)$$

Port feed Suppose a guided wave \vec{E}_p propagates along positive z axis with propagation constant β . Furthermore, suppose that \vec{E}_p is the only propagating mode at the chosen frequency. The port feed model can be implemented with the Robin boundary condition by choosing

$$\alpha = i\beta \quad \text{and} \quad g = 2i\beta (\vec{n} \times \vec{E}_p) \times \vec{n}. \quad (19.11)$$

19.3 Keywords

Constants

The keywords of this are used to change the values of natural constants.

Permittivity of Vacuum Real

The permittivity of vacuum ε_0 . Defaults to $8.854187817 \cdot 10^{-12} \frac{\text{C}}{\text{Vm}}$.

Permeability of Vacuum Real

The permeability of vacuum μ_0 . Defaults to $4\pi \cdot 10^{-7} \frac{\text{Vs}}{\text{Am}}$.

Material material id

The material parameters ε_r and μ_r^{-1} are defined in this section.

Relative Permittivity Real

The real part of relative permittivity ε_r .

Relative Permittivity im Real

The imaginary part of relative permittivity ε_r .

Inverse Relative Permeability Real

The real part of μ_r^{-1} .

Inverse Relative Permeability im Real

The imaginary part of μ_r^{-1} .

Inverse Permeability Real

The real part of μ^{-1} .

Inverse Permeability im Real

The imaginary part of μ^{-1} .

Reluctivity Real

The real part of μ^{-1} .

Reluctivity im Real

The imaginary part of μ^{-1} .

Relative Reluctivity Real

The real part of μ_r^{-1} .

Relative Reluctivity im Real

The imaginary part of μ_r^{-1} .

Note that reluctivity and inverse permeability are equivalent quantities here.

Keywords for VectorHelmholtzSolver

Solver solver id

The solver section defines equation solver control variables. Most of the possible keywords – related to linear algebra (starting with Linear System), for example – are common for all the solvers and are explained elsewhere.

Equation String

A string identifying the solver. This can be changed but it must be given. Example:

VectorHelmholtzSolver

Procedure File "VectorHelmholtz" "VectorHelmholtzSolver"

Name of the solver subroutine.

Variable String

The identifier for the field variable to be solved. Real and imaginary parts must be provided, e.g., E[E re:1 E im:1].

Angular Frequency Real

The angular frequency ω .

Use Piola Transform Logical

Utilize modern Piola transformed edge elements. Increases number of DOFs on meshes containing hexahedral and pyramidal elements. If mesh contains elements that are not affine images of the reference element then this option should be enabled.

Linear System Preconditioning Damp Coefficient Real

If present, the preconditioner is constructed from damped system matrix $A + \kappa(S - M + B)$, where A is the original total system matrix, S is the stiffness matrix containing the curl terms, M is the mass matrix, B is the matrix arising from boundary integrals and κ is the damping coefficient.

Linear System Preconditioning Damp Coefficient im Real

Imaginary part of the damping coefficient κ .

Lossless cavity models usually benefit from shifted preconditioning. Utilizing BiCGStab(l) with the ILU(0) or Vanka preconditioner and choosing damping coefficient $\kappa = i$ is likely to result in a convergent iteration.

Body Force `bf id`

The impressed current \vec{J} is specified in the Body Force section.

Current Density `i Real`

The i :th component of the real part of the current density \vec{J} given by $\tilde{\vec{J}} = i\omega\vec{J}$.

Current Density `im i Real`

The i :th component of the imaginary part of the current density \vec{J} given by $\tilde{\vec{J}} = i\omega\vec{J}$.

Boundary Condition `bc id`

`E re {e} Real`

Real part of the DOF value for edge basis functions. Setting this and the three below to 0 guarantees homogeneous Dirichlet boundary conditions.

`E im {e} Real`

Imaginary part of the DOF value for edge basis functions.

`E re {f} Real`

Real part of the DOF value for face basis functions.

`E im {f} Real`

Imaginary part of the DOF value face basis functions.

Electric Robin Coefficient `Real`

Real part of α .

Electric Robin Coefficient `im Real`

Imaginary part of α .

Magnetic Boundary Load `i Real`

The i :th component of real part of g .

Magnetic Boundary Load `i im Real`

The i :th component of imaginary part of g .

Keywords for VectorHelmholtzCalcFields

Solver `solver id`

Unique id for the solver.

Equation `String`

A string identifying the solver.

Procedure `File "VectorHelmholtz" "VectorHelmholtzCalcFields"`

The name of the post processing subroutine.

Angular Frequency `Real`

The angular frequency ω .

Calculate Elemental Fields `Logical`

Calculate elementwise constant approximation of the fields. Useful for discontinuous material parameters.

Calculate Magnetic Field Strength `Logical`

Output magnetic field strength \vec{H} .

Calculate Magnetic Flux Density `Logical`

Output magnetic flux density \vec{B} .

Calculate Poynting Vector `Logical`

Output the Poynting vector $\frac{1}{2}\vec{E} \times \vec{H}^*$.

Calculate Div of Poynting Vector Logical
 Output the divergence of Poynting vector

$$\nabla \cdot \frac{1}{2} \vec{E} \times \vec{H}^* = \frac{1}{2} i \omega \left(\mu^{-1} \vec{H} \cdot \vec{H}^* - \vec{E} \cdot (\epsilon \vec{E})^* \right) - \frac{1}{2} \vec{E} \cdot \vec{J}^*.$$

Also generates result variable Electric Work corresponding to the term $\vec{E} \cdot \vec{J}^*$ above.

Calculate Electric field Logical
 Output the electric field \vec{E} .

Calculate Energy Functional Logical
 Evaluate the left hand side of (19.6) with the discrete field solution.

Bibliography

- [1] J.D. Jackson. *Classical Electrodynamics*. John Wiley & Sons, third edition, 1999.
- [2] A.F. Peterson. Absorbing boundary conditions for the vector wave equation. *Microwave and Optical Technology Letters*, 1(2):62–64, 1988.

Model 20

Computation of Magnetic Fields in 2D

Module name: MagnetoDynamics2D

Module subroutines: MagnetoDynamics2D, MagnetoDynamics2DHarmonic, BSolver

Module authors: Juha Ruokolainen, Eelis Takala, Mika Malinen, Peter Råback

Module status: Mature

Document authors: Peter Råback, Mika Malinen

Document created: 21.1.2013

Document edited: 27.5.2019

20.1 Introduction

This module may be used to solve a version of the Maxwell equations in the 2D (and axially symmetric) special cases when the unknown is the z-component (or ϕ -component) of the vector potential. In contrast to the 3D version of the magnetodynamics solver described earlier the standard Lagrange interpolation is here applied. In addition to performing the computations in the time domain, the analogous version of the equations may also be solved in the frequency domain. Furthermore, an additional solver may be called to produce the magnetic field intensity from the computed vector potential. Also Joule losses may be computed for harmonic fields.

20.2 Theory

When the current density acts in a direction orthogonal to the plane considered, the effect of the scalar potential vanishes in the A-V formulation of Maxwell's equations. The system is then fully described by the vector potential $\vec{A} \equiv A_z \vec{e}_z$ as

$$\sigma \frac{\partial A_z}{\partial t} \vec{e}_z + \nabla \times \left(\frac{1}{\mu} \nabla \times A_z \vec{e}_z \right) = J_z \vec{e}_z + \nabla \times \vec{M} \quad (20.1)$$

where J_z is the current density and \vec{M} is a planar magnetization vector satisfying $\vec{M} \cdot \vec{e}_z = 0$.

The harmonic version of the equation is obtained by replacing the operator $\frac{\partial}{\partial t}$ with $i\omega$. For the harmonic case the Joule heat generation in conductors may be computed from

$$h = \frac{1}{2} \sigma \omega^2 |A_z|^2.$$

As the electric conductivity σ is discontinuous over material boundaries it is attractive to compute a field without it so that the multiplication is carried out within the heat solver where the source term is needed.

20.2.1 Boundary Conditions

For the equation one can apply either Dirichlet or homogeneous natural boundary conditions. The Dirichlet boundary condition for A_z is simply

$$A_z = A_z^b. \quad (20.2)$$

Alternatively flux conditions may be used,

$$\frac{1}{\mu} \frac{\partial A_z}{\partial n} = g. \quad (20.3)$$

It may be difficult to extend the Dirichlet conditions far enough. Then a spherically symmetric far-field approximation may be used resulting to a boundary condition of Robin kind for the flux

$$g = \frac{1}{\mu} A_z \frac{\vec{r} \cdot \vec{n}}{r^2}. \quad (20.4)$$

If no boundary conditions are specified, the natural boundary condition with $g = 0$ prevails.

20.3 Keywords

Keywords for MagnetoDynamics2D

Here we list the keywords that are relevant to solving the evolutionary and stationary versions of the equations by calling the solver subroutine `MagnetoDynamics2D` and that may also be common to the other solvers. Such common keywords relate to specifying material parameters, body forces, and boundary conditions.

Constants

`Permeability of Vacuum` `Real`

This constant has the default value $4\pi \cdot 10^{-7}$ in SI units. In different unit system change this accordingly.

Material `mat id`

The following material parameters may be used by all the solvers in the module.

`Electric Conductivity` `Real`

This keyword is used to specify the electric conductivity σ .

`Relative Permeability` `Real`

If this keyword is used, the permeability μ can be specified in terms of the permeability of vacuum. To obtain the permeability, the value of this keyword is then internally multiplied with the permeability of vacuum. Instead of using this keyword, the keywords `Permeability` or `Reluctivity` may be used.

`Permeability` `Real`

This keyword may be used to specify directly the permeability μ .

`Reluctivity` `Real`

The value of this keyword specifies the reluctivity ν . The permeability is then taken to be $\mu = 1/\nu$.

`Magnetization i` `Real`

The components of the magnetization vector, $i = 1, 2$.

`H-B Curve` `Cubic` `Real`

The $H - B$ curve must be given as a cubic spline. This enables that the derivative of the curve is computed analytically from the spline coefficients.

Solver `solver id`

Equation String MgDyn2D

A describing name for the discrete model handled by this solver may be given by using this keyword. The name can be changed as long as it is used consistently.

Procedure File "Magnetodynamics2D" "Magnetodynamics2D"

This declaration specifies the name of the solver subroutine.

Variable String Az

The name of the variable may be freely chosen provided it is used consistently also elsewhere. The associated number of degrees of freedom should always be one.

Nonlinear System Max Iterations Integer

If the material laws are nonlinear the equation may need some iterations before reaching the solution. This keywords gives the maximum number of iterations. The default is one. If a nonlinear $H - B$ curve is given then Newton's linearization is applied after the 1st iteration.

Nonlinear System Convergence Tolerance Real

This keyword gives the convergence tolerance for the nonlinear iteration.

Body Force bf id

In the body force section the user may give various volume sources.

Current Density Real

This keyword is used to specify the current density in the z-direction.

Boundary Condition bc id

Az Real

This keyword is used to specify the Dirichlet condition for the vector potential.

Infinity BC Logical

Sets far-field conditions for the vector potential assuming spherical symmetry at distance.

Mortar BC Integer

This enforces continuity in the case of rotating boundary conditions by the mortar finite element method. Note that this feature is still under development.

Keywords for Magnetodynamics2DHarmonic

Here only the additional keywords related to the harmonic solver are listed. For other keywords see the definitions above.

Material mat id

Electric Conductivity im Real

Imaginary part of the electric conductivity.

Magnetization i Im Real

Imaginary components of the Magnetization vector, $i = 1, 2$.

Solver solver id

Equation String MgDyn2DHarmonic

A name for the solver.

Procedure File "Magnetodynamics2D" "Magnetodynamics2DHarmonic"

This declaration specifies the name of the solver subroutine.

Variable String Potential[Potential Re:1 Potential Im:1]

The name of the variable may be freely chosen provided it is used consistently also elsewhere. The associated number of degrees of freedom should always be two.

Body Force `bf id`

In the body force section the user may give various volume sources.

Current Density `Im Real`

This keyword is used to specify the imaginary part of the current density.

Keywords for Bsolver

An additional solver may finally be called to compute derived fields.

Solver `solver id`

The postprocessing solver currently only solves for the magnetic field density. The size of the requested vector field is 2 when the target variable is real-valued and 4 if it is complex-valued. The user does not need to specify the output fields.

Equation `String BSolver`

A describing name for the solver is given. This can be changed as long as it is used consistently.

Procedure `File "MagnetoDynamics2D" "BSolver"`

The name of the solver subroutine is given.

Target Variable `String`

This keyword is used to specify the name of the underlying potential variable, the default is `Az`.

Discontinuous Galerkin `Logical`

The derived fields are discontinuous if the material properties has jumps. Therefore the visualizations are more appealing if the fields may be allowed to be discontinuous. Setting this flag `True` activates discontinuous Galerkin (DG) computation of the fields. Note that these fields are compatible only with certain postprocessing practices. One possible way is to use `vtu` output and ask elemental fields for saving, such as `Vector Field Elemental 1`.

Average Within Materials `Logical`

If DG formulation for the fields is asked, this enforces averaging of the fields within materials.

Calculate Joule Heating `Logical [True]`

In large computations the automatic computation of the Joule heating may be turned off by this keyword. The default is `False`. The keyword is only applicable for the harmonic case. The computation results to two additional variables. `Joule Heating` gives the absolute heating and `Joule Field` the field that gives the heating when multiplied by the electric conductivity. This may be needed if the electric conductivity is discontinuous making also the heating power discontinuous.

Desired Heating Power `Real`

A constant that gives the desired total heating power in Watts. If the keyword is active, then the `Joule Heating` and `Joule Field` are multiplied by the ratio of the desired and computed heating powers.

Model 21

Magnetic Induction Equation

Module name: MagneticSolve

Module subroutines: MagneticSolver

Module authors: Juha Ruokolainen

Document authors: Ville Savolainen, Antti Pursula

Document edited: May 24th 2005

21.1 Introduction

The magnetic induction equation describes interaction of a conducting liquid or gas with applied and induced magnetic fields in the low-frequency domain. The induction equation for the magnetic flux density is always coupled to the Navier-Stokes equation for the movement of the fluid. The magnetic field, in turn, causes the Lorentz force in the Navier-Stokes equation. The fluid is typically hot, and the Navier-Stokes equation is often coupled also to the heat equation.

The induction equation solver can also be used in a body without a moving fluid, i.e., when $\vec{v} = 0$ and the Navier-Stokes equation is not solved. In this case, the problem belongs to the field of magneto-quasistatics.

21.2 Theory

The magnetic induction equation may be derived from the Maxwell's equations, with the displacement current in Ampère's law neglected, and the Ohm's law for conducting fluids, $\vec{j} = \sigma(\vec{E} + \vec{v} \times \vec{B})$. This approximation for the behavior of electromagnetic fields in conducting, moving fluids is called magnetohydrodynamics.

The magnetic induction equation is given by

$$\frac{\partial \vec{B}}{\partial t} + \frac{1}{\sigma \mu} \nabla \times \nabla \times \vec{B} - \nabla \times (\vec{v} \times \vec{B}) = 0, \quad (21.1)$$

where σ is the electric conductivity and μ the magnetic permeability of the material. These must be specified by using the keywords `Electric Conductivity` and `Magnetic Permeability` in the `Material` section.

The force term induced by the magnetic field for the flow momentum equations is given by

$$\vec{f}_m = \vec{j} \times \vec{B}, \quad (21.2)$$

and the Joule heating in the heat equation by

$$h_m = \frac{1}{\sigma} |\vec{j}|^2, \quad (21.3)$$

where \vec{j} is the current density, calculated from the Ampère's law $\vec{j} = \nabla \times \vec{H}$. These body forces are specified by the keywords `Lorentz Force` and `Joule Heat`.

The magnetic field can also be divided into external, or applied, and induced field, $\vec{B} = \vec{B}^e + \vec{B}^i$. The external magnetic field \vec{B}^e is created by permanent magnets or currents outside the fluid. The external field may be given to the induction equation solver either from a restart file, e.g., as calculated by the magnetostatic solver, or defined via the `sif` file's keywords `Applied Magnetic Field 1, 2 and 3`. If the restart file is used, the components of \vec{B}^e are read from the variables named `magnetic flux density 1, 2 and 3`. If both methods are used, the two applied fields are summed together. It is assumed that the sources of the external field are outside the flow region, i.e., $\nabla \times \vec{B}^e = 0$, and that the time derivative of the external field can be ignored. The time derivative $\partial \vec{B}^e / \partial t$ can, however, be specified directly by the keywords `Magnetic Bodyforce 1, 2 and 3`. The induction equation solver gives the components of the induced magnetic field \vec{B}^i .

Both transient and steady-state solvers for the magnetohydrodynamical system (induction, Navier-Stokes and heat equations) are available. The magnetostatic and time-harmonic solvers for the external magnetic field are described elsewhere in the Models Manual. In some cases it is also possible that the velocity is *a priori* known, for example when studying induction in a rotating body. Then a user defined velocity can be used instead of computing the velocity from Navier-Stokes equations.

Currently the induction equation can be solved in a cylindrically symmetric or a general three-dimensional formulation.

21.2.1 Boundary Conditions

For the induction equation one can apply either Dirichlet or natural boundary conditions. In both cases, one must check that the computational domain is extended far enough to avoid numerical errors. For this reason, it is possible to solve the magneto-quasistatics problem in an adjacent body.

The Dirichlet boundary condition for a component of the induced magnetic field B_i (we have dropped now the superscript i that marked the induced field) is

$$B_i = B_i^b. \quad (21.4)$$

B_i^b can be a constant or a function of time, position or other variables. The keywords for the Dirichlet boundary conditions are `Magnetic Field 1, 2 and 3`.

In the cylindrically symmetric case, the Dirichlet boundary condition for the azimuthal component B_ϕ is in the same units as for the other two components, i.e., in T, and not for a contravariant component. On the symmetry axis one has to set $B_r = 0$ and $B_\phi = 0$, and $\partial B_z / \partial r = 0$ is applied implicitly.

If no Dirichlet condition is specified, natural boundary condition is applied.

21.3 Keywords

`Solver solver id`

Note that all the keywords related to linear solver (starting with `Linear System`) may be used in this solver as well. They are defined elsewhere.

`Equation String [Magnetic Induction]`

The name of the equation. It is also possible to use this solver as external procedure. Then the name of the equation must not be the above (use e.g. `Magnetic Field Solver`). Also the following four keywords have to be added with the values give here.

`Procedure File "MagneticSolve" "MagneticSolver"`

`Variable String Magnetic Field`

`Variable DOFs Integer 3`

`Exported Variable 1 = -dofs 3 electric current`

The above four keywords are to be given only when using the solver as an external procedure.

Nonlinear System Convergence Tolerance *Real*

This keyword gives a criterion to terminate the nonlinear iteration after the relative change of the norm of the field variable between two consecutive iterations k is small enough

$$\|\vec{B}^k - \vec{B}^{k-1}\| < \epsilon \|\vec{B}^k\|,$$

where ϵ is the value given with this keyword.

Nonlinear System Max Iterations *Integer*

The maximum number of nonlinear iterations the solver is allowed to do. If neither the material parameters nor the boundary conditions are functions of the solution, the problem is linear, and this should be set to 1.

Nonlinear System Relaxation Factor *Real*

Giving this keyword triggers the use of relaxation in the nonlinear equation solver. Using a factor below unity is sometimes required to achieve convergence of the nonlinear system. A factor above unity might speed up the convergence. Relaxed variable is defined as follows:

$$\vec{B}' = \lambda \vec{B}^k + (1 - \lambda) \vec{B}^{k-1},$$

where λ is the factor given with this keyword. The default value for the relaxation factor is unity.

Steady State Convergence Tolerance *Real*

With this keyword a equation specific steady state or coupled system convergence tolerance is given. All the active equation solvers must meet their own tolerances for their variable u , before the whole system is deemed converged. The tolerance criterion is:

$$\|u_i - u_{i-1}\| < \epsilon \|u_i\|,$$

where ϵ is the value given with this keyword.

Equation *eq id*

The equation section is used to define a set of equations for a body or set of bodies:

Magnetic Induction *Logical*

If set to True, solve the magnetic induction equation.

User Defined Velocity *Logical*

Controls whether the velocity is given by the user or computed by another solver. Default value is False, which means that velocity solution of Navier-Stokes equations is used.

Navier-Stokes *Logical*

If set to True, solve also the Navier-Stokes equations. For magnetohydrodynamics, this is done, except when the computational region for the magnetic field is extended beyond the fluid.

Heat Equation *Logical*

If set to True, solve also the heat equation.

Body Force *bf id*

The body force section may be used to give additional force terms for the equations.

Lorentz Force *Logical*

If set true, triggers the magnetic field force for the flow momentum equations.

Joule Heat *Logical*

If set true, the Joule heating is added in the heat equation.

Magnetic Bodyforce *i Real*

This keyword can be used to specify explicitly the time dependence of the external field, i.e., the term $-\partial \vec{B}^e / \partial t$. This is especially useful for time-harmonic fields, where the time derivative can be calculated and expressed easily.

Initial Condition `ic id`

The initial condition section may be used to set initial values for the field variables. The following variables are active:

Magnetic Field `i Real`

For each magnetic flux density component $i = 1, 2, 3$.

Material `mat id`

The material section is used to give the material parameter values. The following material parameters may be set for the induction equation. They can be a constant or a function of a given variable.

Magnetic Permeability `Real`

The magnetic permeability is set with this keyword. For most fluids, the vacuum value for μ_0 can be used, and the keyword set to $1.25664\text{e-}6$.

Electric Conductivity `Real`

The value of the electric conductivity is set with the keyword. For example, for polythermal flows the conductivity could be a function of the temperature.

Applied Magnetic Field `i Real`

This keyword can be used to specify the external field, or a part of it, and its contribution to the term $\nabla \times (\vec{v} \times \vec{B}^e)$. The field may be a function of, e.g., time or position.

MHD Velocity `i Real`

The user defined velocity can be given with these keywords with $i = 1, 2, 3$.

Boundary Condition `bc id`

The boundary condition section holds the parameter values for various boundary condition types. Dirichlet boundary conditions may be set for all the primary field variables. The ones related to induction equation are

Magnetic Field `i Real`

Dirichlet boundary condition for each magnetic flux density component $i = 1, 2, 3$.

Model 22

Reduced Dimensional Electrostatics

Module name: StatElecBoundary

Module subroutines: StatElecBoundaryForce, StatElecBoundaryEnergy, StatElecBoundaryCharge, StatElecBoundarySpring

Module status: Beta

Module authors: Peter Råback

Document authors: Peter Råback

Document created: 18 Feb 2002

Document edited: 2 Jan 2011

22.1 Introduction

In some applications the geometry is such that the 3D electrostatics may quite accurately be reduced to a 1D problem. This is the case for nearly aligned planes. If the angle between the planes is φ (in radians) the error of this approximation is roughly $2\varphi^2/3$. Therefore we may use an analytical solution that results directly from the distance of the planes that are in different potential. The ideal model may be further developed by taking into account perforated structures and dielectric layers.

22.2 Theory

It is assumed here that the electric field is stationary in the time-scale under study. The electric field \vec{E} may be expressed with an electric scalar potential ϕ ,

$$\vec{E} = -\nabla\phi. \quad (22.1)$$

If there are no free charges, the scalar potential may be solved from

$$-\nabla \cdot \epsilon \nabla \phi = 0. \quad (22.2)$$

When one dimension is much smaller than the other two we may assume that the field is one-dimensional. Then the electric field resulting from potential difference $\Phi = \Delta\phi$ is

$$\vec{E} = E\vec{n} = \frac{\Phi}{d}\vec{n}, \quad (22.3)$$

where \vec{n} is the unit normal and $d(\vec{r})$ is the height of the aperture. The energy density per unit area is now,

$$e = \frac{1}{2}\epsilon E^2 d = \frac{\epsilon \Phi^2}{2d}. \quad (22.4)$$

which corresponds to a induced charge density on the surface

$$q = \frac{\varepsilon\Phi}{d}. \quad (22.5)$$

The force is obtained from the derivative of the energy,

$$f = \frac{\partial e}{\partial d} = -\frac{\varepsilon\Phi^2}{2d^2}, \quad (22.6)$$

and the spring constant from the derivative of the force,

$$k = \frac{\partial f}{\partial d} = \frac{\varepsilon\Phi^2}{d^3}, \quad (22.7)$$

The forces and spring constants are always aligned in the direction of the surface normal since any other direction is incompatible with the original assumptions.

22.2.1 Electrostatics of perforated structures

If there are holes or other imperfections in the structure they may be homogenized over the whole area. By computing the electric energy and force in the presence and absence of holes we get correction factors

$$e_{holes} = \alpha e_{ideal} \quad (22.8)$$

and

$$f_{holes} = \beta f_{ideal} \quad (22.9)$$

The correction terms may be precalculated for a given geometry. However, if the relative change in the aperture is large the correction terms should be modeled in some manner. we would also like to have similar expressions for the spring constant

$$k_{holes} = \gamma k_{ideal}. \quad (22.10)$$

If we assume that e_{holes} is proportional to $1/d$ then the following relations may easily be derived.

$$\beta = \alpha - \alpha' d \quad (22.11)$$

and

$$\gamma = \alpha - \alpha' d + \frac{1}{2} \alpha'' d^2, \quad (22.12)$$

where the derivation is done respect to d .

Now we are only left with the problem of finding a nice functional approximation for α . The holes in the membrane may be expressed using three dimensionless variables $\tilde{d} = d/r$, $\tilde{b} = b/r$ and $\tilde{R} = R/r$. Here r is the hole radius, b the hole depth, d the aperture and R the distance between holes. When $\tilde{R} \gg 1$ and $b \gg 1$ the correction depends only on \tilde{d} .

Numerical computations suggest that the correction $\alpha(\tilde{d})$ should approach unity as the distance \tilde{d} approaches unity. On the other hand, it should approach $1 - q$ for small values of d . Here q is the area fraction of the holes.

Numerical calculations suggest that a second order rational polynom gives quite an accurate fit to the computed results,

$$\alpha(d) = 1 - q \frac{1}{1 + a_1 d + a_2 d^2}. \quad (22.13)$$

Fully analytical formulas are now more tedious but the values for β and γ are easily calculated using the derivatives

$$\alpha(d)' = q \frac{a_1 + 2a_2 d}{(1 + a_1 d + a_2 d^2)^2}, \quad (22.14)$$

and

$$\alpha(d)'' = 2q \frac{a_2 - 2a_1^2 - 3a_1 a_2 d - 3a_2^2 d^2}{(1 + a_1 d + a_2 d^2)^3}. \quad (22.15)$$

Least squares fitting to the numerical computations suggest that for cylindrical hole $a_1 = 4.2523$, $a_2 = 0.4133$, for a rectangular slot $a_1 = 2.3198$, $a_2 = 0.2284$ and for a square hole $a_1 = 3.8434$, $a_2 = 0.3148$. When fitting the model the suggested constant term diverged up to 4 % from unity but the value one was enforced anyway because it has the nice limiting value properties.

22.2.2 Dielectric layer

If the conductor is covered with a dielectric layer we need to modify the equations. We assume that the aperture consists of two materials with permittivities ε_1 and ε_2 and thicknesses d_1 and d_2 . Because the flux must be the same this means that the fields are

$$E_1 = \frac{\Phi}{d_1 + \varepsilon_1 d_2 / \varepsilon_2} \quad (22.16)$$

and

$$E_2 = \frac{\Phi}{\varepsilon_2 d_1 / \varepsilon_1 + d_2}. \quad (22.17)$$

Defining $d_x = d_1 + \varepsilon_1 d_2 / \varepsilon_2$ these become

$$E_1 = \frac{\Phi}{d_x} \quad (22.18)$$

and

$$E_2 = \frac{\varepsilon_1}{\varepsilon_2} \frac{\Phi}{d_x}. \quad (22.19)$$

The total energy density is then

$$e = \frac{1}{2} \varepsilon_1 \Phi^2 \frac{d_1}{d_x^2} + \frac{1}{2} \frac{\varepsilon_1^2}{\varepsilon_2} \Phi^2 \frac{d_2}{d_x^2} = \frac{\varepsilon_1 \Phi^2}{2 d_x}. \quad (22.20)$$

We assume that the resonator moves so that d_1 changes and d_2 remains constant. Then the force density is

$$f = \frac{\partial e}{\partial d_1} = \frac{\partial e}{\partial d_x} \frac{\partial d_x}{\partial d_1} = -\frac{\varepsilon_1 \Phi^2}{2 d_x^2}. \quad (22.21)$$

And similarly the spring constant density

$$k = \frac{\partial f}{\partial d_1} = \frac{\varepsilon_1 \Phi^2}{d_x^3}. \quad (22.22)$$

These expressions may be used inside the integral instead of the constant field values to account for the dielectric layer. It may be noted that the equations are exactly the same as for the case without the layer except that the aperture d is replaced with the efficient aperture $d_x = d_1 + \varepsilon_1 d_2 / \varepsilon_2$.

22.3 Implementation issues

This module is not a solver in itself. It only provides boundary conditions for real models. Natural models to combine with these boundary conditions are models describing deformation in solid structures. For plates the conditions are applied to the leading dimension while for generic 3D solids the conditions are applied to the boundaries. Therefore the same subroutines may be applied to either boundary or to material section. There is actually just one subroutine and the value it returns is defined by the name of the routine used to call it.

These routines here were historically developed for MEMS modeling in a different setting and were much later added to the open source publication as a lighter version.

22.4 Keywords

Constants

Permittivity Of Vacuum Real [8.8542d-12]

The default is given in SI units. In other units the constant should be changed appropriately.

Boundary Condition bd id

Procedure "StatElecBoundary" "StatElecBoundaryForce"
Function that returns the nodal force density.

Procedure "StatElecBoundary" "StatElecBoundaryCharge"
Function that returns the nodal charge density.

Procedure "StatElecBoundary" "StatElecBoundaryEnergy"
Function that returns the nodal energy density.

Procedure "StatElecBoundary" "StatElecBoundarySpring"
Function that returns the nodal spring density.

Gap Height Real
Distance on which the 1D electrostatic model is applied for. May depend on displacement, for example, via MATC functions.

Potential Difference Real
Potential difference between the plates.

Relative Permittivity Real
Relative permittivity of the material between the plates.

Layer Thickness Real
There may be a non-conducting layer on top of the plate. If this keyword is not defined no layer is assumed.

Layer Permittivity Real
Relative permittivity of the layer.

Hole Type String [slot / round / square]
The 1D electrostatics can account also for perforated structures if the depth of the hole is large compared to the width of the hole. The different hole geometries are an infinite slot, a round hole and a square hole.

Hole Size Real
The size of the hole is for a round hole the radius, for a square half the side and for a slot half of the width.

Hole Fraction Real
The fraction of the holes on the surface.

Hole Depth Real
The depth of the holes i.e. also the thickness of the perforated plate.

Model 23

Poisson-Boltzmann Equation

Module name: PoissonBoltzmannSolve

Module subroutines: PoissonBoltzmannSolve

Module authors: Peter Råback

Document authors: Peter Råback

Document edited: 10.8.2004

23.1 Introduction

The macroscopic electromagnetic theory is governed by the Maxwell's equations. In steady state the electric field may usually be solved from a simple Poisson equation. However, if there are free charges in the domain that are affected by the electric field the equation is no longer valid. Also the contribution of the free charges need to be taken into consideration. If the electrostatic force is the only force affecting the distribution of the electric charges then the potential in the steady-state is given by the Poisson-Boltzmann equation [1]. This equation may find its use in microfluidics and electrochemical applications. Note that if the charge distribution is affected by the flow distribution of the carrier fluid this equation is no longer valid.

23.2 Theory

The electrostatic equation for the electric potential ϕ yields,

$$-\nabla \cdot \varepsilon \nabla \phi = \rho, \quad (23.1)$$

where ε is the permittivity of the medium and ρ is the charge density. Assuming that there is a fixed charge density and both positive or negative moving ions the charge may be written as

$$\rho = \rho_0 + e(z^- n^- + z^+ n^+) \quad (23.2)$$

where ρ_0 is interior charge distribution of fixed positions of all solute charges, and e is the unit charge of a electron, and z is the charge number of the positive or negative ions, and n is the corresponding ion density.

The electrochemical potential μ of the ions is defined by $\mu = ez\phi + k_B T \ln n$, where the first term is the electrostatic contribution and the second term comes from the entropy of the ions at the weak solution limit. In equilibrium μ_i is constant over the whole domain and thus the ion density obeys a Boltzmann distribution,

$$n = n_0 e^{-ez\phi/k_B T} \quad (23.3)$$

where k_B is the Boltzmann constant. Inserting this to the Poisson equation we obtain the Poisson-Boltzmann equation that determines the potential field self-consistently,

$$-\nabla \cdot \varepsilon \nabla \phi = \rho_0 + ez^- n_0^- e^{-ez^- \phi/k_B T} + ez^+ n_0^+ e^{-ez^+ \phi/k_B T}. \quad (23.4)$$

A special case of the equation is obtained if the charge numbers and the concentrations are equal, $z = -z^- = z^+$ and $n_0 = n_0^- = n_0^+$. Then the equation simplifies to

$$-\nabla \cdot \varepsilon \nabla \phi = \rho_0 - 2ezn_0 \sinh(ez\phi/k_B T). \quad (23.5)$$

The Poisson-Boltzmann equation is obviously nonlinear. We will show the iterative procedure only for this case, the generic case is dealt similarly.

23.2.1 Iteration scheme

Defining $\alpha = 2ezn_0$ and $\beta = ez/k_B T$ the Poisson-Boltzmann equation for a symmetric electrolyte may be written as

$$-\nabla \cdot \varepsilon \nabla \phi = \rho_0 - \alpha \sinh(\beta \phi). \quad (23.6)$$

The straight-forward iterative procedure treats only the left-hand-side of the equation in an implicit manner,

$$-\nabla \cdot \varepsilon \nabla \phi^{(n+1)} = \rho_0 - \alpha \sinh(\beta \phi^{(n)}). \quad (23.7)$$

The convergence of this scheme is, however, quite poor for many cases of practical interest. An improved strategy should linearize also the right-hand-side.

Making a Taylor's expansion we may approximate

$$\sinh(\beta \phi^{(n+1)}) \approx \sinh(\beta \phi^{(n)}) + \beta \cosh(\beta \phi^{(n)})(\phi^{(n+1)} - \phi^{(n)}) \quad (23.8)$$

which results to the Newton iteration scheme

$$\begin{aligned} & \left[-\nabla \cdot \varepsilon \nabla + \alpha \beta \cosh(\beta \phi^{(n)}) \right] \phi^{(n+1)} \\ &= \rho_0 - \alpha \sinh(\beta \phi^{(n)}) + \alpha \beta \cosh(\beta \phi^{(n)}) \phi^{(n)}. \end{aligned} \quad (23.9)$$

This scheme has good convergence properties and is usually the method of choice.

23.2.2 Boundary conditions

For electric potential either Dirichlet or Neumann boundary condition can be used. The Dirichlet boundary condition gives the value of the potential on specified boundaries. The Neumann boundary condition is used to give a flux condition on specified boundaries

$$\sigma = \varepsilon \nabla \phi \cdot \vec{n}, \quad (23.10)$$

where σ is the surface charge density.

23.2.3 Derived quantities

When the potential has been solved the electric field may be obtained as a postprocessing step from

$$\vec{E} = -\nabla \phi. \quad (23.11)$$

Charge density may be obtained as the right-hand-side of the Poisson equation,

$$\rho = \rho_0 + ez^- n_0^- e^{-ez^- \phi/k_B T} + ez^+ n_0^+ e^{-ez^+ \phi/k_B T}. \quad (23.12)$$

which in symmetric case yields,

$$\rho = \rho_0 - 2ezn_0 \sinh(ez\phi/k_B T). \quad (23.13)$$

The energy density of the field may be computed from

$$e = \frac{1}{2} \vec{E} \cdot \vec{D} = \frac{1}{2} \varepsilon (\nabla \phi)^2. \quad (23.14)$$

However, in a more generic treatment also the contribution of the concentration should be included in the expression of the energy.

23.3 Notes on output control

The user can control which derived quantities (*i.e.* electric field and electric energy) are calculated.

There are also available two choices of visualization types for the derived quantities. The node values can be calculated by taking the average of the derived values on neighboring elements (constant weights). This results often in visually good images. The other possible choice is to weight the average with the size of the elements, which is more accurate and should be used when some other variable depends on these derived values. The latter choice is also the default.

23.4 Keywords

Constants

Permittivity Of Vacuum Real [8.8542e-12 C²/Nm²]

Boltzmann Constant Real [1.3807e-23 J/K]

Unit Charge Real [1.602e-19 C]

Equation equation id

Calculate Electric Energy Logical [False]

Controls whether the electric energy density is written in results files (default False).

Solver solver id

Equation String Poisson Boltzmann Solver

Variable String Potential

This may be of any name as far as it is used consistently also elsewhere.

Variable DOFs Integer 1

Degrees of freedom for the potential.

Procedure File PoissonBoltzmannSolve PoissonBoltzmannSolve

Following are listed three keywords with default values for output control.

Nonlinear System Max Iterations Integer

The maximum number of nonlinear iterations.

Nonlinear System Convergence Tolerance Real

The relative error after which the iteration is terminated.

Nonlinear System Newton After Iterations Integer

The number of iterations after which Newton iteration is turned on. The default is zero which should usually be optimal.

Nonlinear System Newton After Tolerance Real

Optional parameter which gives the tolerance in error after which Newton iteration is turned on.

Calculate Electric Field Logical [True]

Calculate Electric Flux Logical [True]

Constant Weights Logical [True]

Used to turn constant weighting on for the results.

Material mat id

Relative Permittivity Real

The total permittivity is the product of the relative permittivity and the permittivity of vacuum.

Reference Temperature Real

This keyword is used to give the temperature occurring in the Boltzmann factor.

Charge Number Integer

For symmetric cases the charge number. For unsymmetric cases one may give separately Positive Charge Number and Negative Charge Number.

Ion Density Integer

For symmetric cases the original density of ions. For unsymmetric cases one may give separately Positive Ion Density and Negative Ion Density.

An alternative set of parameters are also possible which are particularly suitable for testing purposes. These are limited to the symmetric case where the potential normalized with the Zeta potential is solved. Then the permittivities should be set to unity and only two variables are needed to define the case.

Poisson Boltzmann Beta Real

This keyword gives the ratio of parameter β to the the Zeta potential.

Poisson Boltzmann Alpha Real

This keyword gives the parameter α

Body Force bodyforce id

Charge Density Real

The fixed charge distribution that is not affected by the electric field.

Boundary Condition bc id

Potential Real

Electric Flux BC Logical

Must be set to True if flux BC is used.

Surface Charge Real

Gives the surface charge for the Neumann boundary condition.

Bibliography

- [1] D. Andelman. *Handbook of Biological Physics*, chapter 12. Electrostatic Properties of Membranes: The Poisson-Boltzmann Theory. Elsevier Science, 1995.

Model 24

Loss Estimation Using the Fourier Series

Module names: FourierLoss

Module subroutines: FourierLossSolver

Module authors: Peter Råback, Mika Malinen

Document authors: Mika Malinen, Peter Råback

Document created: 25.5.2013

Document edited: 21.11.2018

24.1 Introduction

The primary motivation for this solver is the estimation of electromagnetic losses by using the Steinmetz equation approach. It could have other uses as well. The main idea is to make a Fourier transformation on-the-fly and compute losses that are proportional to the frequency always when full cycle has been completed.

Given an evolutionary finite element field

$$\vec{A}_h(x, t) = \sum_{j=1}^N \alpha_j(t) \vec{\psi}_j(x), \quad (24.1)$$

the solver enables to replace the evolution of the scalar degrees of freedom $\alpha_j(t)$ for $t \in [t_0, t_0 + T]$ by the Fourier series approximation

$$\alpha_j(t) \approx a_j^0 + \sum_{k=1}^K a_j^k \cos[k\omega(t - t_0)] + \sum_{k=1}^K b_j^k \sin[k\omega(t - t_0)], \quad (24.2)$$

where the angular frequency ω may be defined in terms of a period T as $\omega = 2\pi/T$. The coefficients a_j^k and b_j^k are given by

$$\begin{aligned} a_j^0 &= \frac{1}{T} \int_0^T \alpha_j(t' + t_0) dt', \\ a_j^k &= \frac{1}{T} \int_0^T \alpha_j(t' + t_0) \cos(2\pi k t' / T) dt', \\ b_j^k &= \frac{1}{T} \int_0^T \alpha_j(t' + t_0) \sin(2\pi k t' / T) dt'. \end{aligned} \quad (24.3)$$

In practice the time stepping algorithm gives the values of $\alpha_j(t)$ at only a discrete set of time values and linear interpolation is applied to generate $\alpha_j(t)$ at the other points.

The use of (24.2) in (24.1) now yields

$$\vec{A}_h(x, t) \approx \vec{a}_0(x) + \sum_{k=1}^K \vec{a}_k(x) \cos[k\omega(t - t_0)] + \sum_{k=1}^K \vec{b}_k(x) \sin[k\omega(t - t_0)] \quad (24.4)$$

where the fields $\vec{a}_k(x)$ and $\vec{b}_k(x)$ have the finite element expansions

$$\vec{a}_k(x) = \sum_{j=1}^N a_j^k \vec{\psi}_j(x) \quad \text{and} \quad \vec{b}_k(x) = \sum_{j=1}^N b_j^k \vec{\psi}_j(x).$$

If the field of interest is $\vec{B} = \nabla \times \vec{A}_h$, we have similarly

$$\vec{B}(x, t) \approx \nabla \times \vec{a}_0(x) + \sum_{k=1}^K \nabla \times \vec{a}_k(x) \cos[k\omega(t - t_0)] + \sum_{k=1}^K \nabla \times \vec{b}_k(x) \sin[k\omega(t - t_0)]. \quad (24.5)$$

If the problem setup is given in 3D case it is assumed that the solution is obtained using edge element basis. In the special case of 2D target field and the target variable is expected to be a scalar field $A_h(x, t)$ and \vec{B} is then generated as

$$\vec{B} = \frac{\partial A_h}{\partial y} \vec{e}_x - \frac{\partial A_h}{\partial x} \vec{e}_y.$$

24.2 Loss estimation

In a typical application we have in mind the field \vec{A}_h is taken to be the vector potential solution corresponding to the AV formulation of electromagnetic equations. Then the field $\vec{B} = \nabla \times \vec{A}_h$ gives the magnetic flux density which, in view of (24.5), may be approximated in the form

$$\vec{B}(x, t) \approx B_0(x) + \sum_{k=1}^K B_k(x) \cos[k\omega(t - t_0) - \phi_k],$$

with ϕ_k a phase angle.

The loss power associated with each simple-harmonic component may then be estimated over a body Ω by using the Steinmetz equation approach as

$$P_k = \int_{\Omega} C f_k^{\alpha} B_k^{\beta} d\Omega \quad (24.6)$$

where C , α and β are given data and $f_k = k\omega/(2\pi)$. The total loss P is then obtained as

$$P = \sum_{k=1}^K P_k. \quad (24.7)$$

The field variable P_h associated with this solver is the total loss power distribution per unit volume which is obtained from the weak formulation

$$\int_{\Omega} P_h v_h d\Omega = \int_{\Omega} \sum_{k=1}^K C f_k^{\alpha} B_k^{\beta} v_h d\Omega \quad (24.8)$$

where v_h denotes a suitable test function. The current implementation enables several terms with constant exponents α and β . The coefficient C , on the other hand varies among materials and may be a function of frequency.

24.3 Keywords

Solver `solver id`

Procedure `File "FourierLoss" "FourierLossSolver"`

This keyword is used to give the Elmer solver the place where to search for the routine producing the loss estimate.

Equation `String [FourierLoss]`

A name to the computational version of the loss equation may be given by using this keyword. The name has no effect but it should be unique.

Target Variable `String`

The value of this keyword gives the name of the field for which the Fourier series expansion is produced. It is usually assumed that the field is magnetic vector potential. It may be either nodal (with 1 component in 2D and 3 components in 3D) or a curl-conforming vector field.

Target Variable AV `Logical`

The user may enforce the target variable to be one resulting from the AV-solver which uses the edge degrees of freedom to express the vector potential. If the flag is not given, the existence of the AV solver is deduced from the size of the permutation vector. Because the detection may be errourness, for example when a DG field is used, this may also be given value `False` to enforce use of a nodal field.

Target Variable Direct `Logical`

By default it is assumed that the target variable is the magnetic vector potential and the curl operator is applied. However, with this flag it is possible to give either scalar field $|B|$ or vector field \vec{B} directly. The handicap of this is that the field is not continuous. However, this may be particularly usefull for testing purposes.

Variable `String`

The primary variable is the loss component for the linear frequency dependence. The default name is `Fourier Loss` if all components are summed together and `Fourier Loss i` if each component is treated separately.

Separate Loss Components `Logical`

By setting this flag to `True` the user may save the different loss components in Steinmetz model as separate fields for postprocessing. Otherwise they will be merged to one single field.

Inexact Integration `Logical`

The integration may be performed most accurately by integrating the product of the linear approximation of the time-dependence and the sines and cosines analytically. Currently this exact integration is used. However, the user may replace it with inexact version even though this is not recommended.

Simpsons Rule `Logical`

If using inexact integration, the default method is the trapezoidal integration. Instead the user may request the Simpson's rule for better accuracy. If exact integration is used, this keyword has no effect.

Discontinuous Galerkin `Logical`

Instead of the standard Galerkin formulation the user may also choose to compute the post-processed fields using discontinuous Galerkin (DG) approximation. This enables discontinuities between elements. However, the cost is often quite large since the matrix structures related to DG are very large compared to the standard Galerkin method. The lumped quantities are computed on-the-fly so for them there is no difference.

Average Within Materials `Logical`

This only applies when the DG method is used to compute the loss fields. When this flag is turned on the fields within same material are weakly enforced to be continuous. This results to much smoother fields while still maintaining the discontinuity over material interfaces.

Calculate Elemental Fields *Logical*

The user may request elemental fields to be computed. These are natural since they allow discontinuous fields to be visualized. This features uses the same degrees of freedom as the DG approximation but does not include interactions between elements.

Calculate Nodal Losses *Logical*

This may be used to calculate the losses directly in terms of power (i.e. Watts in SI units) lost in each node. For conforming meshes this provides the easiest coupling with heat equation and these losses become directly the r.h.s. source terms for the heat equation. Nodal losses are always lumped to one field even though the distributed fields would be separated.

Fourier Series Components *Integer*

This keyword is used to define the parameter K in (24.2), i.e. how many Fourier series components are generated.

Fourier Series Output *Logical*

If the value `True` is given, then the Fourier component fields \vec{a}_k and \vec{b}_k are output into the result file. Note that for edge elements this does not currently work as they cannot be directly written to a file but they should be mapped into a space that is able to be visualized.

Angular Frequency *Real*

This keyword is used to give the angular frequency ω . Alternatively, the simulation section may be used for the same purpose.

Frequency *Real*

Instead of giving the angular frequency, the user may specify the frequency $f = \omega/(2\pi)$ by using this keyword.

Fourier Start Time *Real*

This keyword can be used to define the start time t_0 for performing the integration.

Fourier Start Timestep *Integer*

This keyword can be used to define the start time t_0 such that $t_0 = t_n$, with n the timestep index given by using this keyword.

Fourier Start Cycles *Integer*

This keyword can be used to define the start time t_0 such that $t_0 = nT$, with n the value of this keyword.

Fourier Integrate Cycles *Integer*

By default the Fourier coefficient computation is restarted after integration over one complete period T . If this keyword is used, then the restarting occurs after integration over n periods, with n the value of this keyword. The reason for this could be that the results include some randomness that we would like to filter out by integrating over several cycles.

Harmonic Loss Frequency Exponent (K) *Real*

The value of this keyword gives the parameters α_k in a vector format. The number of components, K , should be the same as the number terms in Steinmetz loss model. Alternatively the user may give this componentwise, i.e. `Harmonic Loss Frequency Exponent 1`, etc.

Harmonic Loss Field Exponent (K) *Real*

The value of this keyword gives the parameters β_k in a vector format. The number of components, K , should be the same as the number terms in Steinmetz loss model. Alternatively the user may give this componentwise, i.e. `Harmonic Loss Field Exponent 1`, etc.

Fourier Loss Filename *File*

Name for the file in which the losses will be saved. Losses will be saved so that each body for which there are any losses is written to a separate line. If the name is not given, nothing will be saved.

Material *material id***Harmonic Loss Coefficient i** *Real*

This keyword is used to define the material parameter C in (24.6) for each simple-harmonic term

- i.* The highest existing value i determines the number of terms in the loss model. Note that the coefficient may only be a function of frequency itself.

Model 25

Coil Current Solver

Module name: CoilSolver
Module subroutines: CoilSolvrr
Module authors: Peter Råback
Module status: Beta
Document authors: Peter Råback
Document edited: 16.5.2017

25.1 Introduction

Assume that we have been given a geometry of a coil. Now the coil consists of individual wires and therefore the direction of the current is basically well defined. Unfortunately it is often difficult to provide the directional information of the wires. In simple cases (such as ideal cylinder) the user may give the current in the coil using analytical functions. In general case this is however, not possible.

This solver tries to provide numerical means for setting the coil current. The idea is that the computed current density (or potential) may then be used in further simulations of magnetic fields.

Now the natural way to create the currents is to solve a static current conduction equation. Unfortunately for closed loops it is difficult to define the boundary conditions. Rather than trying to make some special mesh this solver provides a unique strategy to create the currents in two pieces using Dirichlet conditions for some nodes in the bulk.

25.2 Theory

Our starting point for defining currents in the coil is to use an equation for the static current conduction,

$$-\nabla \cdot \sigma \nabla \phi = 0. \quad (25.1)$$

where ϕ is the electric potential, and σ is the electric conductivity. For this equation we may either set Dirichlet boundary conditions

$$\phi = \phi_0 \quad (25.2)$$

or Neumann boundary conditions

$$-\sigma \frac{\partial \phi}{\partial n} = j_n. \quad (25.3)$$

If we solve the equation with the standard Galerkin method in FEM the volume current density may then be calculated using the same finite element spaces from

$$\vec{j} = -\sigma \nabla \phi. \quad (25.4)$$

Normalization of potential and current

Typically we know the total current over the cross section

$$J = \int j_n d\Gamma. \quad (25.5)$$

If the current density over the boundary area A is constant this gives the current density, $j_n = J_0/A$. Often it is more ideal to define Dirichlet conditions and normalize the potential afterwards to give the desired total current,

$$\phi := \frac{J_0}{J} \phi \quad (25.6)$$

and similarly for the current,

$$\vec{j} := \frac{J_0}{J} \vec{j}. \quad (25.7)$$

If we know that the cross section of the coil is constant and thereby also the current density is constant j_0 we may normalize the current density also locally taking only the direction from the initial current,

$$\vec{j} := j_0 \frac{\vec{j}}{|\vec{j}|}. \quad (25.8)$$

Now this normalization should not be done light-heartedly since after this the solution might not be divergence-free any more. When the magnitude of the solution is known *a priori* we could use also other means of obtaining the direction field.

Modified conductivity field

Now in the previous we assumed that conduction is isotropic and homogeneous. Unfortunately this means that for coils the current won't follow the wires as it tends to take the shortest path more easily. On the other hand we don't know the anisotropic conductivity since if we knew it we would also know the direction of the current and there would be nothing to compute. Therefore we look at some ways to iterative determine the solution from the already computed field.

One idea is to introduce an additional field variable, say c , such that

$$-\nabla \cdot c \sigma \nabla \phi = 0. \quad (25.9)$$

$$c |\sigma \nabla \phi| = j_0. \quad (25.10)$$

This may be iteratively solved by solving ϕ from the 1st equation and c from the 2nd equation. Unfortunately numerical tests showed that the resulting field for c that is prone to numerical oscillations. Therefore we add some regularization to the 2nd equation by adding some diffusion

$$-\nabla \cdot D \nabla c + |\sigma \nabla \phi| c = j_0. \quad (25.11)$$

where D is the numerical diffusion. Unfortunately the equation does not necessarily have any solution. In fact real tests showed that increasing value of D will make the solution smoother but at the same time it will be more difficult to find a solution that would fulfil both equations. Still the current density may be closer to the real current density than in the homogeneous case. Just a small number of iterations will get the most of the benefits of the scaled conductivity field.

We may also look at the conductivity in an anisotropic form. One idea is to use the knowledge that in the true coil the conductivity is only in the direction of the gradient. We may therefore look at the conductivity in the form

$$\sigma = \sigma_0 \frac{\nabla \phi}{|\nabla \phi|} \quad (25.12)$$

Now without the multiplier c this would not lead to an improvement because then the direction does not have an effect as the flux is always in the direction of the gradient only. However, the combined effect of these two could improve the accuracy of the direction of the current density.

There can be also partially geometric means for setting the direction of the conductivity field. Basically the distance from the coil surface, s , defines a field the gradient of which defines a direction with zero conductivity. So we take a isotropic conductivity as a starting point and eliminate the conductivity in the direction of the normal i.e.

$$\sigma = \sigma_0 \frac{1}{\sqrt{3}} [I - \text{abs}(\nabla s)] \quad (25.13)$$

Unfortunately this formulation has some difficulties at the center of the coil where the the distance function is poorly defined.

For structured and extruded meshes one could also use the directions of the element edges to directly set the direction of the current. Starting from a rough information for the current direction the edges aligned with the current could be detected and thereafter the current direction could be refined accurately with the geometry. This method has not been implemented.

Case of closed coils

The closed coils are dealt in two parts. The idea is to use rough fiction boundary conditions for the bulk nodes. Setting potentials to 0 and ϕ_0 on a narrow gap around the fictitious interface we can generate a current to the coil. Now this current is "good" only in the other side of the coil where the effects of the poorly defined boundary conditions have settled. The distance from the fictitious interface should be at least around three times the diameter of the coil cross section. Therefore the approach is better suited for "lean" coils. We apply the same procedure to opposite sides of the coil and normalize the coil current to be the same. The final current is an union of the two cases so that we always take the more accurate of the two computed currents fields.

Now the currents should ideally be continuous. However, the potential derived in this way can never be continuous since at some stage the potential should have a jump over the coil. Still, if the coil potential is used only element-wise so that it is always operated by a gradient we may use a special subroutine that returns always the more accurate set of nodal potentials for each element. After taking the gradient of the potentials the solution should be continuous.

Using the coil solver

The solver can be used to compute currents or potentials to be used by other solvers. The standard case could equally well be computed by the `StatCurrentSolver` but this solver includes some ways to make the current distribution more accurate to the case of coils. Unfortunately all these techniques are based on heuristics and they will most likely offer better current distributions than the standard approach.

If one tries to normalize the current density to a predefined constant value the user should be sure that the cross section of the coil is constant. Otherwise the user introduces a current source or drain that may be difficult to treat in the later steps of the simulation.

When using in conjunction with the `MagnetoDynamics` module it is advisable to perform the fixing of the current to be divergence free using the internal `Jfix` solver. Otherwise the solution might not exits for the equation.

Currently it is assumed that there is just one coil, and for the closed coil it is assumed that the coil axis is in the direction of the z -axis. These limitations are not inherent to the method and could be resolved by small coding effort.

25.3 Keywords

The solver includes some internal definitions which eliminates some keywords. For example, the variable related to the solver is internally created and also the boundary conditions related to it are set internally so the user does not need to know of it.

```
Solver solver id
```

```
Equation String CoilSolver
```

Procedure File "CoilSolver" "CoilSolver"

Name of the solver module and solver.

Coil Closed Logical

Is the coil closed. If it is then the the potential will be computed in two parts. Note that for many coils it is currently assumed that all the coils are either open or closed. Closed coils will in effect create a secondary potential field and an auxiliary field `PotSelect` used to toggle between the two potentials.

Coil Conductivity Fix Logical

Fix the coil conductivity so that the current density would be more even.

Cfix diffusion D

iffusion coefficient for regularization of c field.

Coil Anisotropic Logical

Make the conductivity be aligned with the gradient of the potential.

Calculate Coil Current Logical [True]

Calculate the current flowing in the coil(s). The effect of the enforced currents in the coil may be given as a source term either as current, or as a enforced potential.

Use Wall Distance Logical

Use wall distance to introduce anisotropy into the coil conductivity. If wall distance is used then it is assumed that a field `Wall Distance` exists.

Save Coil Set Logical

Optionally save the `CoilSet` field.

Save Coil Index Logical

Optionally save the `CoilIndex` field. This makes sense only if there are more than one coil. Then the index gives the number of the coil.

Normalize Coil Current Logical

After the current has been computed normalize it to the desired magnitude if this flag is given.

Nonlinear System Max Iterations Integer

For the inhomogeneous cases give the number of iterations. The default is 2 for the scaled conductivity, and 3 for the scaled anisotropic conductivity.

The following keywords define one single coil. They may be located in the `Solver` section if there is just one single coil. If there are many coils they should be located in different `Component` sections.

Desired Coil Current Real

The desired coil current J_0 in the coil. The default is 1.

Desired Current Density Real

The desired coil current density j_0 in the coil. The default is 1.

Coil Cross Section Real

Cross section (area) of the coil that may be used to related total current and current density.

Coil Center(3) Real

Center of the coil in (x, y, z) cartesian coordinates. If coil center is not given then the volumetric mid point is assumed to be the center.

Coil Normal(3) Real

The user may give the coil axis around which the coil circulates around. If not given the coil axis is found as the axis that gives the maximum inertial momentum for the coil. The tangent directions are deduced from the coil normal.

Coil Bandwidth Real

A parameter related to the width of the fictitious interface used to set the potentials in an automated way. This is relative to the total width. Default is 20% of the width of the coil.

Narrow Interface Logical

This flag enforces the use of narrow strategy for the setting of Dirichlet conditions and computation of the resulting nodal charges. It is a better strategy than the wider coil bandwidth strategy particularly for thick coils. Makes the above keyword obsolete.

Boundary Condition bc id

Potential Real

Dirichlet BC for the potential.

Current Density BC Logical

Must be set to True if Neumann BC is used.

Coil Start Logical

Defines a boundary where coil starts. Not needed if coil is closed.

Coil End Logical

Defines a boundary where coil ends. Not needed if coil is closed.

The user can use the united potential also for closed coils by using a special subroutine. The calling convention would then be, for example, in the MagnetoDynamics module in the Body Force section

```
Electric Potential = Variable time
```

```
Real Procedure "CoilSolver" "CoilPotential"
```

Here time is just a dummy variable. Similarly for coil potential normalized on-the-fly within each element,

```
Electric Potential = Variable time
```

```
Real Procedure "CoilSolver" "CoilPotentialNormalized"
```

Part V

Other Physical Models

Model 26

Electrokinetics

Module name: Electrokinetics

Module subroutines: helmholtz_smoluchowski1, helmholtz_smoluchowski2, helmholtz_smoluchowski3, helmholtz_smoluchowski

Module author: Thomas Zwinger

Document author: Thomas Zwinger

Document created: April 13th 2005

26.1 Introduction

If dealing with electrolytic fluids constrained to small volumes, surface forces caused by electric surface charges in combination with externally applied electrostatic fields are sufficient strong to affect the fluid volume. If these effects are utilized to attenuate the fluid volume, we talk of *Electrokinetics*. The term *Electroosmotic Flow* (EOF) is used in connection with the attenuation of a net charge inside a originally neutral electrolyte caused by separation induced by a surface charge of a wall.

26.2 Theory

Chemical reactions between the contents of a liquid and the wall material may lead to a net charge of the containment at the wall-liquid interface. If the liquid is an electrolyte (i.e., it contains free ions), ions of opposite charge align along the wall creating the *Stern layer*. Adjacent to the Stern layer, a charge separation - called the *diffuse layer* of the initially neutral electrolyte takes place. Due to the two layer structure the whole area of charge separation in the vicinity of a wall is called the *Electric Double layer* (EDL).

26.2.1 Electroosmotic slip velocity

Considering a symmetric electrolyte – i.e., the bulk ion density of ions with opposite valence numbers $\pm z$ are equal $n_0^+ = n_0^- = n_0$ – at a certain temperature, T , the typical width-scale of the EDL is given by the *Debye length* [1]

$$\lambda_D = \left(\frac{\epsilon_f \epsilon_0 k_b T_0}{2 n_0 z^2 e_0^2} \right)^{1/2}. \quad (26.1)$$

Here e_0 stands for the unit charge and k_b denotes the Boltzmann constant. The relative permittivity of the electrolyte and the permittivity of vacuum are given by ϵ_f and ϵ_0 , respectively.

The potential, Φ and the volume charge density, ρ_e , within the EDL are tightly coupled to each other by the Poisson-Boltzmann equation (23.4) (see chapter 23). In order to exactly resolve the dynamics close to the walls, (23.4) should be solved and the resulting specific electric force then be considered in the equation of motion. Nevertheless, provided the typical length scales of the flow perpendicular to the containment walls, H , strongly exceed those of the EDL – in other words, we obtain very small values for the non-dimensional

group $\mathcal{L} = \lambda_D/H \ll 1$ – the dynamics of the electrolyte inside the EDL does not have to be resolved at all. In this case simple considerations of a force balance between shear stress and electric force lead to a slip condition for the fluid [2]. At the boundary, the tangential velocity is set to the *Helmholtz-Smoluchowski* velocity

$$\vec{u}_{\text{tang.}} = \vec{u}_{\text{H-S}} = \frac{\vec{E}_{\text{tang.}} \epsilon_f \epsilon_0 \zeta}{\mu_f}, \quad (26.2)$$

with μ_f standing for the local fluid viscosity. The *zeta potential*, ζ – a property depending on the electric properties of the wall material as well as the electrolyte – usually is determined experimentally. From a physical point of view it can be interpreted as the value of the solution obtained by (23.4) at the Stern layer. The tangential component, $\vec{E}_{\text{tang.}}$, of the external electric field, \vec{E} , is evaluated from the outward pointing surface normal \vec{n} , applying the following relation

$$\vec{E}_{\text{tang.}} = \vec{E} - (\vec{E} \cdot \vec{n}) \vec{n} \quad (26.3)$$

Alternatively, the resulting slip velocity may be related to the tangential field using the *Electroosmotic Mobility*, μ_{EO}

$$\vec{u}_{\text{H-S}} = \mu_{\text{EO}} \vec{E}_{\text{tang.}}. \quad (26.4)$$

A combination of (26.2) and (26.4) leads to the following identity

$$\mu_{\text{EO}} = \frac{\epsilon_f \epsilon_0 \zeta}{\mu_f}. \quad (26.5)$$

26.3 Limitations

- The Helmholtz-Smoluchowski velocity should not be applied if the non-dimensional group \mathcal{L} defined in 26.2.1 is of unity order or larger. Then the potential- and charge density distribution as well as the dynamics of the electrolyte inside the EDL has to be resolved.
- In a strict sense, the Helmholtz-Smoluchowski theory applies only to configurations where the normal-component of the external field, $\vec{E} \cdot \vec{n}$, is small. If dealing with electric insulating wall materials – as it is usually the case in microfluidic applications – this condition is implicitly complied with.
- The assumption of a Newtonian fluid underlies the derivation of the Helmholtz-Smoluchowski velocity.
- The function `helmholtz_smoluchowski` can only be applied on boundaries of two-dimensional domains, where the tangential direction is uniquely defined.

26.4 Keywords

Keywords for `helmholtz_smoluchowski`

Constants

Permittivity Of Vacuum Real [8.8542e-12 C²/Nm²]
 permittivity of vacuum, only needed if Helmholtz-Smoluchowski velocity is defined using expression (26.2)

Equation `equation id`

Electric Field String [computed, constant]
 the option for how to evaluate the electric field should be set to one of these values.
 If set to `computed`, the function will search for Electric Field {1,2,3} in the list of solver variables. If set to `constant`, the function will search for Electric Field {1,2,3} in the section `Material material id`, where `material id` is the id-number associated with the material parameter list of the electrolyte

Material `material id`

If the Helmholtz-Smoluchowski velocity is defined using expression (26.2), then the following keywords have to be provided in this section

Viscosity `Real`
viscosity of the electrolyte

Density `Real`
volumetric density of the electrolyte

Relative Permittivity `Real`
relative permittivity of the electrolyte

Boundary Condition `bc id`

In two-dimensional configurations the Helmholtz-Smoluchowski velocity directly can be assigned to the tangential component of the velocity field

Normal Tangential Velocity `Logical True`
Velocity 2 = Variable `Dummyargument`
`Real Procedure "Electrokinetics" "helmholtz_smoluchowski"`
Sets tangential EO slip velocity

The argument `Dummyargument` can be any existing variable, since it is not used to evaluate the velocity.

In three-dimensional configurations (and as an alternative also in two-dimensional), the velocity has to be defined for each component

Normal Tangential Velocity `Logical False`
Velocity 1 = Variable `Dummyargument`
`Real Procedure "Electrokinetics" "helmholtz_smoluchowski1"`
Velocity 2 = Variable `Dummyargument`
`Real Procedure "Electrokinetics" "helmholtz_smoluchowski2"`
Velocity 3 = Variable `Dummyargument`
`Real Procedure "Electrokinetics" "helmholtz_smoluchowski3"`

The argument `Dummyargument` can be any existing variable, since it is not used to evaluate the velocity.

If the Helmholtz-Smoluchowski velocity is defined using expression (26.2), then the zeta potential, ζ , for the specific boundary region has to be defined

Zeta Potential `Real`
Sets the zeta-potential for this boundary

Alternatively, the user can declare the EO-mobility, as explained in (26.5)

EO Mobility `Real`
Sets EO mobility for this boundary

Bibliography

- [1] G.E. Karniadakis and A. Beskok. *Micro flows : fundamentals and simulation*. Springer-Verlag, New York, Berlin, Heidelberg, 2001.
- [2] R.-J. Yang, L.-M. Fu, and Y.-C. Lin. Electroosmotic Flow in Microchannels. *J. Colloid and Interface Science*, 239:98–105, November 2001.

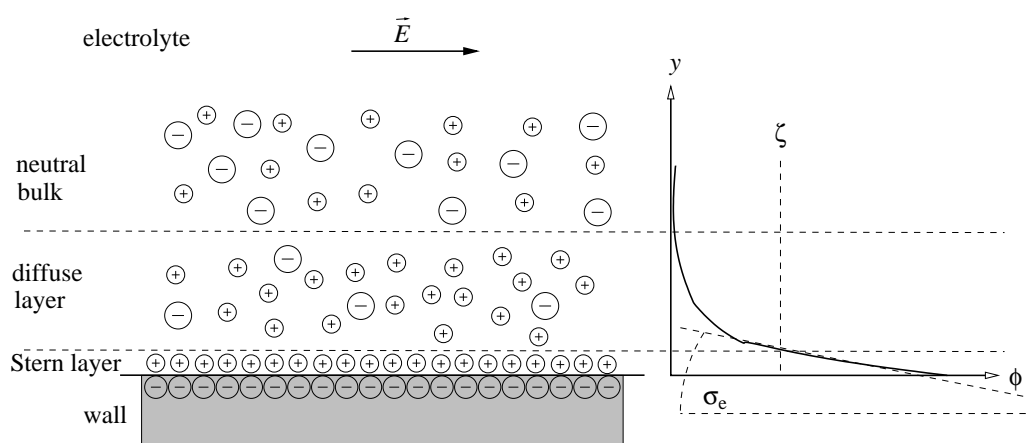


Figure 26.1: Structure of the EDL. The value of the induced potential, Φ at the Stern layer usually is referred to as the zeta-potential, ζ

Model 27

Block Preconditioning for the Steady State Navier–Stokes Equations

Module name: ParStokes, PressurePrecond, VelocityPrecond

Module subroutines: StokesSolver, PressurePrecond, VelocityPrecond

Module authors: Mika Malinen, Jonas Thies, Juha Ruokolainen

Document authors: Mika Malinen

Document edited: Dec 18, 2015

27.1 Introduction

The discretization of the incompressible Navier–Stokes equations usually leads to large linear systems which cannot be solved efficiently with the standard iterative methods. Special preconditioning strategies should therefore be employed to obtain rapid convergence of iterations. In this section we describe a special solver for the steady state Navier–Stokes equations which has utility when the Reynolds number is moderate. The development of the solver was originally motivated by needs to solve the full Stokes equations in connection with glaciological simulations. Therefore, the possibility to utilize parallel computation has been in mind from the very beginning (the module name comes from *Parallel Stokes* solver).

The speciality of the solver is that it contains an in-built two-level iteration scheme to handle the linear systems arising from the linearization and discretization. Inner iterations can be associated with preconditioning and they provide search directions for the outer iterative method (GCR) applied to the primitive problem. The preconditioning strategy is here based on the idea of block preconditioning via utilizing the natural block structure of the discrete system. In practice simpler auxiliary problems need to be solved in order to find the update directions for the velocity and pressure unknowns in a decoupled manner. In this way the door is opened to utilizing other efficient methods, such as multigrid methods for the discrete Poisson problems, in connection with the solution of the more complicated problem.

The alternate flow solver contained in the `ParStokes` module basically mimics the standard Navier–Stokes (NS) solver of Elmer. However, it does not provide all features available in the standard NS solver.

27.2 The model

The equations to be solved are written as

$$\begin{aligned} \rho_0(\mathbf{u} \cdot \nabla)\mathbf{u} - \operatorname{div}[2\mu(\mathbf{D})\mathbf{D}(\mathbf{u})] + \nabla p &= \rho_0\mathbf{g}, \\ -\operatorname{div}\mathbf{u} &= 0 \end{aligned} \tag{27.1}$$

or

$$\begin{aligned} -\operatorname{div}[2\mu(\mathbf{D})\mathbf{D}(\mathbf{u})] + \nabla p &= \rho_0\mathbf{g}, \\ -\operatorname{div}\mathbf{u} &= 0 \end{aligned} \tag{27.2}$$

when the effect of convection is neglected to obtain the Stokes equations. Here $\mathbf{D}(\mathbf{u})$ is the symmetric part of the velocity gradient, the constant ρ_0 is the fluid density and μ is the fluid viscosity. For example, in the case of Glen's flow law we have $\mu(\mathbf{D}) = 1/2A^{-k}[I_2(\mathbf{D})]^{(k-1)/2}$, with A and k parameters and $I_2(\mathbf{D}) = 1/2(\mathbf{D} \cdot \mathbf{D})$.

Common boundary conditions may be expressed as

$$\begin{aligned} \mathbf{u} &= \hat{\mathbf{u}} & \text{on } \Gamma_D, \\ 2\mu(\mathbf{D})\mathbf{D}(\mathbf{u})\mathbf{n} - p\mathbf{n} &= \hat{\mathbf{s}} & \text{on } \Gamma_N, \\ \mathbf{u} \cdot \mathbf{n} &= 0 \quad \text{and} \quad \mathbf{n} \times [2\mu(\mathbf{D})\mathbf{D}(\mathbf{u})\mathbf{n}] \times \mathbf{n} = -\beta\mathbf{n} \times \mathbf{u} \times \mathbf{n} & \text{on } \Gamma_S. \end{aligned} \quad (27.3)$$

Here $\hat{\mathbf{u}}$ is the specified velocity, $\hat{\mathbf{s}}$ is the specified traction, and also the friction coefficient $\beta > 0$ is given as initial data.

27.3 Linearization

Either Newton's method or the strategy of Picard type can be used to linearize the effect of viscosity. The possible convection term is always linearized with the Picard method.

27.4 Discretization aspects

The solver is tailored to the case of the lowest-order continuous pressure approximation. Equal-order approximations of (\mathbf{u}, p) are unstable, but the solver offers the following two strategies to obtain stable methods:

- *P_2 - P_1 / Q_2 - Q_1 approximation.* The polynomial order of the velocity approximation can be taken to be of the second order. This option requires that the finite element mesh contains second-order elements based on the Lagrange interpolation.
- *A hierarchic version of bubble-stabilized methods.* The lowest-order velocity approximation may also be enhanced relative to the pressure by using elementwise bubble functions. The richness of velocity approximation depends on how many bubble basis functions are constructed. For example, the set of basis functions for the linear triangular and tetrahedral elements can be augmented with one interior bubble function by giving the element type definition `Element = "p:1 b:1"` in the Equation section. Analogous rectangular and brick elements may also be constructed, but our experience is that more than one bubble function may be necessary to obtain stability, making this strategy less attractive. It is recommended that the definition `Element = "p:1 b:4"` is generally used in three dimensional cases. The Q_2 - Q_1 and P_2 - P_1 approximation methods may require less computational work, especially for three-dimensional problems where the burden of numerical integration in the assembly phase is increased significantly.

It is noted that other instabilities generally arise when the flow is convection-dominated. The computational methods of the solver have not been designed to handle this scenario, i.e. the Reynolds number for the flow should be moderate.

27.5 Block preconditioning

The linearization and discretization of the flow model considered leads to solving linear systems of the form

$$\begin{bmatrix} A_k & B^T \\ B & 0 \end{bmatrix} \begin{bmatrix} U \\ P \end{bmatrix} = \begin{bmatrix} F \\ 0 \end{bmatrix}, \quad (27.4)$$

where A_k is the coefficient matrix for the velocity unknowns at the nonlinear iteration step $k + 1$ and B is the divergence matrix. If the linear system (27.4) is abbreviated as $Kx = b$, the outer iterative method based on the generalized conjugate residual method (GCR) can be used to generate iterates $x^{(k)}$ that minimize the residual 2-norm

$$\|b - Kx^{(k)}\|_2$$

over the search direction space

$$\mathcal{X}_k = x^{(0)} + \text{span}\{s^{(1)}, s^{(2)}, \dots, s^{(k)}\}$$

where $x^{(0)}$ is the initial guess. In this setting, the preconditioner is considered to be an operator P which, given the previous iterate, produces the new search direction $s^{(k+1)}$.

Here the search directions are found as approximate solutions of systems

$$P s^{(k+1)} = b - K x^{(k)}. \quad (27.5)$$

The preconditioner matrix is selected to be of the form

$$P = \begin{bmatrix} T & B^T \\ 0 & Q \end{bmatrix} \quad (27.6)$$

where T approximates A and $Q = \mu^{-1}M$, with M the pressure mass matrix. By default the choice $T = A$ is used. An alternate is to select T to be the block diagonal approximation of A or the scaled vector-Laplacian matrix, which is the discrete version of the operator $-\mu \operatorname{div} \nabla$ subject to suitable boundary conditions.

To apply the preconditioner via performing approximate solves of (27.5), the user must define methods for solving subproblems of the type

$$Q \delta P^{(k+1)} = R_P^{(k)} \quad \text{and} \quad T \delta V^{(k+1)} = R_A^{(k)} - B^T \delta P^{(k+1)}$$

or, in practice, their preconditioned versions

$$(Q P_Q^{-1}) \delta \hat{P}^{(k+1)} = R_P^{(k)} \quad (27.7)$$

and

$$(T P_T^{-1}) \delta \hat{V}^{(k+1)} = R_A^{(k)} - B^T \delta V^{(k+1)}. \quad (27.8)$$

The efficiency of the block-preconditioned solver depends heavily on how these auxiliary problems are solved (the solves associated with the coefficient matrix T are the most critical). A crucial aspect of the methodology is that these subsidiary problems can be considerably easier to solve than the original fully coupled system. They may also be solved inexactly without impairing the performance of the preconditioner. In addition, it is noted that performing highly accurate solutions of the linearized systems (27.4) is not needed in the beginning of the nonlinear iteration when the iterates are not accurate. The solver provides an option to employ adaptive stopping criteria so that the solution accuracy for (27.4) is adapted automatically based on the size of the current nonlinear error.

27.6 Defining additional solvers for subsidiary problems

If the block preconditioner is applied, the solver input file must contain two additional solver sections to enable the assembly of the subsidiary problems with the coefficient matrices T and Q . In this connection special equation names (given as the value of `Equation` keyword) have to be used. If the value of the parameter `d` defines the space dimension, these solver sections should be written as

```
Solver 1
  Equation = "Velocity Preconditioning"
  Procedure = "VelocityPrecond" "VelocityPrecond"
  Variable = "v"
  Variable DOFs = $d
  ! Potential commands to adjust the solution of velocity
  ! preconditioning problems:
  ...
End
```

```

Solver 2
  Equation = "Pressure Preconditioning"
  Procedure = "PressurePrecond" "PressurePrecond"
  Variable = "P"
  Variable DOFs = 1
  ! Potential commands to adjust the solution of pressure
  ! preconditioning problems:
  ...
End

```

The first solver section is needed for creating the velocity preconditioning system (27.8), while the second solver section corresponds to the pressure preconditioning system (27.7). Each of these sections may also contain additional keyword commands to change the default linear solver and its parameters. The default variable names V and P can be changed freely.

If a boundary condition $\mathbf{u} = \hat{\mathbf{u}}$ of Dirichlet type is specified, the variable of the velocity preconditioning equation δV (the Elmer variable V above) must also be constrained similarly by using the homogeneous Dirichlet condition. Usually there is no need to specify boundary conditions for the pressure preconditioning variable.

27.7 Examples

The solver described here has for example been applied to simulate flows of ice sheets. To obtain an example in this field, see the ISMIP HOM A test case

`.../elmerfem/fem/tests/ParStokes_ISMIP_HOM_A010/`

in the source code repository.

27.8 Keywords

Material `material-id`

Density `Real`

This keyword is used to define the density ρ_0 .

Viscosity Model `String`

This keyword can be used to select a more advanced viscosity model such as Glen's law. For available options see the documentation of the standard Navier–Stokes solver.

Viscosity `Real`

This keyword is used to define directly the viscosity μ .

Constant-Viscosity Start `Logical`

In some cases it may be useful to start the nonlinear iteration with a constant viscosity although the intention is eventually to use another model. Starting with a constant viscosity can be avoided by giving the value `False` for this keyword.

Solver `solver-id`

Equation `String`

This keyword declares the name of the equation.

Procedure `File` `"ParStokes"` `"StokesSolver"`

The name of the file and procedure.

Variable `String`

This keyword is used to declare the name of the solution.

Variable DOFs Integer

The value of this keyword defines the number of unknown scalar fields and must hence equal to $d + 1$ where d is the spatial dimensionality of the computational domain. The unknown scalar fields are always numbered in such a way that the highest running number is associated with the pressure solution.

Convective Logical

If the value `True` is given, the convection term will be included so that the steady state version of the incompressible Navier–Stokes equations is solved.

Nonlinear System Convergence Tolerance Real

This keyword defines the stopping criterion for the nonlinear iteration. The nonlinear iteration is terminated when the maximum number of nonlinear iterations is reached or when

$$\|b - K(x)x\|_2 / \|b\|_2 < \varepsilon_N,$$

where ε_N is the value of this keyword.

Nonlinear System Max Iterations Integer

This keyword defines the maximum number of nonlinear iterations.

Nonlinear System Newton After Iterations Integer

If n is the value of this keyword, n Picard updates are performed before switching to Newton's method.

Nonlinear System Newton After Tolerance Real

If the norm of the nonlinear residual is smaller than the value of this keyword, then the nonlinear iteration method is switched to Newton's method.

P2-P1 Approximation Logical

This keyword can be used to select the P_2 - P_1/Q_2 - Q_1 approximation method. The finite element mesh must then contain second-order finite elements based on the Lagrange interpolation.

Element String

If bubble functions are used as the stabilization strategy, then an element definition must be given to specify the number of bubble functions. The recommended choice is "p:1 b:4" for three dimensional simulations.

Block Preconditioning Logical

If the block preconditioner is used, the value of this keyword must be `True`.

Block Diagonal A Logical

This can be used to select the preconditioner matrix T to be the block diagonal approximation of the A -block.

Use Velocity Laplacian Logical

This can be used to select T to be the scaled vector-Laplacian matrix. Then the keyword `Block Diagonal A` must also set to be `True`.

Linear System Convergence Tolerance Real

When the block preconditioning is used, the value of this keyword defines the stopping criterion for the outer GCR method applied to (27.4). The iteration is terminated when

$$\|b - Kx^{(j)}\|_2 / \|b\|_2 < \varepsilon_L,$$

where ε_L is the value of this keyword.

Linear System Adaptive Tolerance Logical

The usage of adaptive stopping criteria can be activated with this keyword.

Linear System Relative Tolerance Real

If the adaptive stopping criteria are employed, this keyword controls the solution accuracy for the linear systems (27.4) during the nonlinear iteration. The stopping tolerance ε_L for (27.4) is chosen to be $\varepsilon_L = \eta_R \cdot \eta_N^{(k)}$, with η_R the value of this keyword and $\eta_N^{(k)}$ the previous nonlinear error. If the value obtained in this way is smaller than the value of the keyword `Linear System Convergence Tolerance`, then this has no effect.

Linear System Base Tolerance Real

The stopping tolerance for solving (27.4) can never be larger than the value of this keyword.

Linear System Max Iterations Integer

When the block preconditioner is used, this keyword is used to define the maximum number of the outer GCR iterations applied to (27.4).

Linear System GCR Restart Integer

The outer GCR iteration can be restarted after m iterations to avoid the increasing cost of the orthogonalization procedure. The value of this keyword specifies the parameter m . The default value is $m = 50$. Giving a larger value can be beneficial if convergence problems relating to the outer iteration are met.

Body Force bf-id

Flow BodyForce i Real

This keyword is used to define the i 's component of the body force vector \mathbf{g} .

Boundary Condition bc-id

Surface Traction i Real

This keyword can be used to specify the components of the traction vector $\hat{\mathbf{s}}$. An alternate for the phrase Surface Traction is Pressure.

Normal Surface Traction Real

This keyword can be used to specify a surface traction in the form $\hat{\mathbf{s}} = \hat{p}\mathbf{n}$ where \hat{p} is the value of this keyword. An alternate for the phrase Normal Surface Traction is External Pressure.

Slip Coefficient i Real

This defines the friction coefficient such that the friction force is proportional to the velocity in the direction i .

Model 28

Rotational Form of the Incompressible Navier–Stokes Equations

Module name: Stokes

Module subroutines: StokesSolver

Module authors: Mika Malinen

Document authors: Mika Malinen

Document edited: June 26th 2009

28.1 Introduction

The basic incompressible flow solver of Elmer uses the standard formulation of the Navier–Stokes equations. This section describes an alternative solver based on the rotational form of the Navier–Stokes system. In addition, some iterative methods that utilize splitting strategies in the solution of the associated discrete problems are represented.

28.2 Field equations

Using the vector identity

$$(\vec{u} \cdot \nabla) \vec{u} = (\nabla \times \vec{u}) \times \vec{u} + \frac{1}{2} \nabla (\vec{u} \cdot \vec{u}), \quad (28.1)$$

the Navier–Stokes system for incompressible Newtonian fluid may be written as

$$\rho \left(\frac{\partial \vec{u}}{\partial t} + (\nabla \times \vec{u}) \times \vec{u} \right) - 2\mu \nabla \cdot \bar{\bar{\epsilon}}(\vec{u}) + \nabla P = \vec{b}, \quad (28.2)$$
$$\nabla \cdot \vec{u} = 0,$$

where $\bar{\bar{\epsilon}}$ is the stretching tensor (2.4) and

$$P = p + \frac{1}{2} \rho \vec{u} \cdot \vec{u} \quad (28.3)$$

is the total (Bernoulli) pressure. The stress $\bar{\bar{\sigma}}$, which may be of interest especially near boundaries, can now be expressed as

$$\bar{\bar{\sigma}} = (-P + \frac{1}{2} \rho \vec{u} \cdot \vec{u}) \bar{\bar{I}} + 2\mu \bar{\bar{\epsilon}}(\vec{u}). \quad (28.4)$$

The system (28.2) provides an alternative starting point for finding discrete solutions. Thus, instead of approximating the conventional primitive variables (\vec{u}, p) , we here look for discrete solutions of (\vec{u}, P) . It should be noted that if the convection term is not taken into account the system (28.2) reduces to the (generalized) Stokes equations. The pressure variable then reduces to the standard pressure, i.e. $P = p$.

28.3 Boundary conditions

Either the normal velocity $\vec{u} \cdot \vec{n}$, with \vec{n} the outward unit normal vector, or the normal surface force $\bar{\sigma} \vec{n} \cdot \vec{n}$ can be prescribed on the boundary. The tangential boundary conditions can be handled systemically in a similar manner. Thus, if \vec{t} is a tangent vector to the boundary, one may prescribe either the tangential velocity $\vec{u} \cdot \vec{t}$ or the tangential surface force $\bar{\sigma} \vec{n} \cdot \vec{t}$.

A rather common way to define an outflow boundary condition for the Navier–Stokes equations is to impose the normal surface force condition

$$\bar{\sigma} \vec{n} \cdot \vec{n} = 0, \quad (28.5)$$

which ensures the uniqueness of the pressure solution. This condition arises when the homogeneous natural boundary condition (do-nothing boundary condition) is imposed in the standard formulation of the Navier–Stokes equations. It should be noted, however, that the homogeneous natural boundary condition associated with the variational formulation of (28.2) can be written as

$$-P\vec{n} + 2\mu\bar{\varepsilon}\vec{n} = \bar{\sigma}\vec{n} - \frac{1}{2}\rho(\vec{u} \cdot \vec{u})\vec{n} = \vec{0}.$$

Thus, a distinction must here be made between the surface force boundary condition and the natural boundary condition. In the case of the rotational form, imposing the homogeneous natural boundary condition in the normal direction yields

$$\bar{\sigma}\vec{n} \cdot \vec{n} = \frac{1}{2}\rho\vec{u} \cdot \vec{u},$$

which, except for the special case of irrotational steady flow of a non-viscous fluid, may be an artificial boundary condition. Nevertheless, the tangential natural boundary condition associated with the rotational form is equivalent to the condition of vanishing tangential surface force, i.e. $\bar{\sigma}\vec{n} \cdot \vec{t} = 0$.

28.4 Linearization

The linearization of the equation of motion in (28.2) can be done by utilizing the Newton iteration. This iteration strategy is based on approximating the rotational convection term as

$$(\nabla \times \vec{u}) \times \vec{u} \approx (\nabla \times \vec{u}) \times \vec{a} + (\nabla \times \vec{a}) \times \vec{u} - (\nabla \times \vec{a}) \times \vec{a}$$

where \vec{a} is the previous velocity iterate. In this connection, the nonlinear boundary condition corresponding to the outflow condition (28.5) is linearized as

$$-P + \frac{1}{2}\rho(2\vec{a} \cdot \vec{u} - \vec{a} \cdot \vec{a}) + 2\mu\bar{\varepsilon}(\vec{u})\vec{n} \cdot \vec{n} = 0.$$

An alternative linearization strategy is to apply Picard's method. Here this method corresponds to linearizing the convection term and the outflow boundary condition as

$$(\nabla \times \vec{u}) \times \vec{u} \approx (\nabla \times \vec{u}) \times \vec{a}$$

and

$$-P + \frac{1}{2}\rho\vec{a} \cdot \vec{u} + 2\mu\bar{\varepsilon}(\vec{u})\vec{n} \cdot \vec{n} = 0.$$

The convergence of the Newton method can be considerably faster than that of Picard's method. Our experience is that this can be the case, especially, when the steady solutions are sought for moderately large Reynolds numbers. However, a difficulty with the Newton method is that the iteration may not be convergent for arbitrary initial guesses. This trouble can often be avoided by performing some Picard updates before switching to Newton's method. In the case of time-accurate simulations this is usually unnecessary since suitably accurate initial guesses are often available from the previous time levels.

28.5 Discretization aspects

The solver is tailored to the case of the lowest-order continuous pressure approximation, but it does not provide any in-built technique to stabilize discrete solutions based on inherently unstable equal-order approximations of (\vec{u}, P) . The solver offers two strategies which can be used to obtain stable methods. First, one can use elements where the velocity approximation is augmented by using elementwise bubble functions. Second, one can utilize hierarchic versions of the second-order elements to rise the polynomial order of the velocity approximation. Both the strategies can be put into effect by utilizing the shape functions for p -elements. Some stable approximation methods are summarized as follows.

- *A hierarchic version of P_2 - P_1 approximation for triangular and tetrahedral elements.* If the basic mesh consists of linear elements (element type 303 or 504), giving the element type definition `Element = "p:2"` in the Equation section switches to the P_2 - P_1 approximation where the velocity approximation is enhanced by using hierarchic basis functions associated with the mid-edge nodes.
- *A hierarchic version of Q_2 - Q_1 approximation for rectangular and brick elements.* Analogously to the previous case, if the basic mesh consists of bilinear or trilinear elements (element type 404 or 808), giving the element type definition `Element = "p:2"` in the Equation section switches to the Q_2 - Q_1 approximation where the velocity approximation is enhanced by using hierarchic basis functions.
- *A hierarchic version of bubble-stabilized methods.* The velocity approximation may also be enhanced relative to the pressure by using elementwise bubble functions. The richness of velocity approximation depends on how many bubble basis functions are constructed. For example, the set of basis functions for the linear triangular and tetrahedral elements can be augmented with one interior bubble function by giving the element type definition `Element = "p:1 b:1"` in the Equation section. Analogous rectangular and brick elements may also be constructed, but our experience is that more than one bubble function may be necessary to obtain stability, making this strategy less attractive. The Q_2 - Q_1 and P_2 - P_1 approximation methods may generally require less computational work, especially for three-dimensional problems where the number of interior bubble functions can be large (notice that in the case of the time-dependent equations the interior degrees of freedom are not eliminated by using the method of static condensation).

It is noted that other instabilities may arise when the flow is convection-dominated. A potentially useful aspect of using the rotational formulation is that, as compared with the standard convection form, instabilities relating to dominating convection may be more benign.

28.6 Utilizing splitting strategies by preconditioning

Discrete Navier–Stokes problems lead usually to large linear systems which are customarily solved with iterative algorithms, in combination with preconditioning. The general preconditioning strategy used in Elmer is based on the computation of incomplete factorizations. The performance of these preconditioners is case-dependent and may not always be satisfactory.

More efficient solution algorithms for a particular problem can often be developed by exploiting the block structure of the linear system. In the following such a solution strategy will be described. Since the application of the preconditioner considered is based on solving certain simpler problems, the door is opened to utilizing other efficient methods, such as multigrid methods for the discrete Poisson problems, in connection with the solution of this more complicated problem.

The linearization and discretization of (28.2) leads to solving linear systems of the form

$$\begin{bmatrix} A & B^T \\ B & 0 \end{bmatrix} \begin{bmatrix} U \\ \Pi \end{bmatrix} = \begin{bmatrix} F \\ 0 \end{bmatrix}, \quad (28.6)$$

where A is the coefficient matrix for the velocity unknowns and B is the divergence matrix. The solution strategy we consider is based on applying a preconditioned Krylov subspace method to (28.6). Given a

previous iterate (U_k, Π_k) , the preconditioning is performed via solving approximately systems of the form

$$\begin{bmatrix} A & 0 & 0 \\ B & M & 0 \\ B & H & S \end{bmatrix} \begin{bmatrix} U_{k+1} - U_k \\ \psi_{k+1} \\ \Pi_{k+1} - \Pi_k \end{bmatrix} = \begin{bmatrix} F - AU_k - B^T \Pi_k \\ -BU_k \\ -BU_k \end{bmatrix}. \quad (28.7)$$

Here M is the pressure mass matrix, while H and S are approximations of (scaled) Laplacian operators. In practice the approximate solution of this block triangular system is generated by applying linear solvers to systems with the coefficient matrices A , M and S . A crucial aspect of the methodology is that these subsidiary problems can be considerably easier to solve than the original fully coupled system. They may also be solved inexactly without impairing the performance of the preconditioner. Moreover, to our experience the performance of the preconditioner is insensitive to discretization parameters and depends only mildly on the Reynolds number, especially in the case of the evolutionary equations. The method is also suitable for finding the steady solutions via using large time step sizes.

The outer iterative method applied to the primary system (28.6) is based on GCR, while the user can specify linear solvers which are used to solve the subsidiary problems related to the preconditioning. It should be noted that boundary conditions associated with the preconditioning operators are built-in, so the user need not specify these constraints.

28.7 Restrictions

Currently, only homogeneous surface force conditions can be imposed on the boundary. If Q_2 - Q_1 or P_2 - P_1 approximation is used, the boundary conditions are set by employing the linear interpolation of boundary data. As a result, optimal accuracy may not be realised. If the preconditioning is done via solving (28.7), the time discretization must be done using BDF(1) and viscosity should be constant.

If decoupled solution strategies are employed, parallel computations are possible only with the version that does not involve performing the outer Krylov iteration update.

28.8 Keywords

Material `material-id`

Density `Real`

This keyword is used to define the density ρ .

Viscosity `Real`

This keyword is used to define the viscosity μ .

Solver `solver-id`

Equation `String`

This keyword declares the name of the equation.

Procedure `File "Stokes" "StokesSolver"`

The name of the file and procedure.

Variable `String`

This keyword is used to declare the name of the solution.

Variable DOFs `Integer`

The value of this keyword defines the number of unknown scalar fields and must hence equal to $d + 1$ where d is the spatial dimensionality of the computational domain. The unknown scalar fields are always numbered in such a way that the highest running number is associated with the pressure solution.

Convective `Logical`

If the value `"False"` is given, the convection term will be neglected so that the generalized Stokes equations are solved.

Nonlinear Iteration Method `String`

This keyword defines the nonlinear iteration method. The default is the Newton method, and Picard's method can be chosen by giving the value "Picard".

Nonlinear System Convergence Tolerance `Real`

This keyword defines the stopping criterion for the nonlinear iteration. The nonlinear iteration is terminated when the maximum number of nonlinear iterations is reached or when

$$\left\| \begin{bmatrix} F - A(U_k)U_k - B^T \Pi_k \\ -BU_k \end{bmatrix} \right\| < TOL \left\| \begin{bmatrix} F \\ 0 \end{bmatrix} \right\|,$$

where TOL is the value of this keyword.

Nonlinear System Max Iterations `Integer`

This keyword defines the maximum number of nonlinear iterations.

Nonlinear System Newton After Iterations `Integer`

If n is the value of this keyword, n Picard updates are performed before switching to Newton's method.

Nonlinear System Newton After Tolerance `Real`

If the norm of the nonlinear residual is smaller than the value of this keyword, then the nonlinear iteration method is switched to Newton's method.

Nonlinear System Relaxation Factor `Real`

If this keyword is used, then the new nonlinear iterate is taken to be

$$(1 - \lambda)(U_k, \Pi_k) + \lambda(U_{k-1}, \Pi_{k-1}),$$

where λ is the value of this keyword.

Block Preconditioning `Logical`

If the block preconditioning via (28.7) is used, the value of this keyword must be "True".

Linear System Convergence Tolerance `Real`

When the block preconditioning is used, the value of this keyword defines the stopping criterion for the outer GCR method applied to (28.6).

Linear System Max Iterations `Integer`

When the block preconditioning is used, this keyword is used to define the maximum number of the outer GCR iterations applied to (28.6). It should be noted that the GCR iteration requires that all previous iterates are saved. Especially in the case of time-accurate simulations the convergence of the preconditioned GCR method is expected to be rapid so that saving all the iterates is not expected to be expensive. If the block preconditioning is used, the solver allocates computer memory based on the value of this keyword, so giving an exaggerated value should be avoided.

Body Force `bf-id`

Body Force i `Real`

This keyword is used to define the i 's component of the body force vector \vec{b} .

Boundary Condition `bc-id`

Outflow boundary `Logical`

If the value "True" is given, then the normal outflow boundary condition (28.5) will be used. Note that this does not define the tangential boundary conditions which have to be specified separately.

If the preconditioning is done via solving (28.7), three additional solver sections need to be written to define linear solvers for subsidiary problems with the coefficient matrices A , M and S . In this connection special equation names (given as values of `Equation` keyword) have to be used. These solver sections should be written as follows.

```

Solver 1
  Equation = "Velocity Preconditioning"
  Procedure = "VelocityPrecond" "VelocityPrecond"
  Exec Solver = "before simulation"
  Variable Output = False
  Variable DOFs = $ d
  Variable = "VelocityCorrection"
  ...
End

Solver 2
  Equation = "Divergence Projection"
  Procedure = "DivProjection" "DivProjection"
  Exec Solver = "before simulation"
  Variable Output = False
  Variable DOFs = 1
  Variable = "DivField"
  ...
End

Solver 3
  Equation = "Pressure Preconditioning"
  Procedure = "PressurePrecond" "PressurePrecond"
  Exec Solver = "before simulation"
  Variable Output = False
  Variable DOFs = 1
  Variable = "PressureCorrection"
  ...
End

```

The first solver section defines a linear solver for the preconditioning system with the velocity matrix A , while the second solver section defines a solver for the system involving the pressure mass matrix M . Finally, the third section is related to the system with the coefficient matrix S arising from the discretization of the Laplacian operator. Each of these sections should also contain the standard keyword commands that actually define the linear solver. Some examples of such definitions can be found in the tests subdirectory of the fem module.

Part VI

**Free Surfaces, Phase Change and
Particle Dynamics**

Model 29

Level-Set Method

Module name: LevelSet

Module subroutines: LevelSetSolver, LevelSetDistance, LevelSetIntegrate, LevelSetCurvature, LevelSetTimestep

Module authors: Peter Råback, Juha Ruokolainen

Document authors: Peter Råback

Document created: 5.4.2006

Document edited: 28.4.2006

29.1 Introduction

There are a number of problems involving free surfaces in continuum mechanics. There are two main strategies to solve them using the finite element method: Lagrangian and Eulerian approach. In the Lagrangian approach the free surface is solved exactly so that it is also an interface between the individual elements. This requires that the computational mesh is distorted in a way that this is possible. However, often the changes in geometry may be too drastic or even the whole topology may change and the Lagrangian approach is no longer feasible. The Eulerian approach describes the interface in a fixed mesh using some additional variable to describe the position of the interface. One possible Eulerian technique is the level-set method (LSM).

In the level-set method the free surface is given as a zero level-set of a higher dimensional variable. E.g. for 2D surfaces the level-set function is defined in 3D space. The level-set function is usually defined to be a signed distance so that inside the domain it obtains a positive value and outside a negative value. The changes in the value of the level-set function mean also that the interface changes the position.

This module includes several different subroutines that may be used when applying the level-set method. Currently there is no reinitialization strategy for 3D problems. Also some other procedures are not fully optimized for the best performance. Therefore the current implementation is best applied to quite simple 2D problems.

29.2 Theory

The interface is defined by a marker function ϕ so that at the interface $\phi = 0$, inside the fluid of interest $\phi > 0$ and elsewhere $\phi < 0$. The interface is update by solving the equation

$$\frac{\partial \phi}{\partial t} + \vec{u} \cdot \nabla \phi = a \quad (29.1)$$

where \vec{u} is the convection field and a is the normal flux on the interface. It is quite challenging to solve the differential equation above without diffusion effects playing a significant role. It is advisable to use 2nd order time-discretization schemes and short timesteps. More precisely, the Courant number $C = |\vec{u}|dt/h$ should be below unity.

It is desirable that the absolute value of function equals the shortest distance to the zero level-set. However, as the level-set function is advected this property may be gradually lost. Therefore a process called reinitialization may be evoked. In 2D the reinitialization may be easily done by geometric procedure. First the zero level-set is formed by going through all the elements and finding the line segments that make the zero level-set. Then the minimum distance of all the nodes is computed by a brute-force search. Assuming there are N nodes and M line segments the search algorithm is $N \times M$ which is quite acceptable complexity for small cases but may become computationally costly in large cases.

The line segments may be assumed to go with the flow and thereby they form an on-the-fly Lagrangian mesh. Therefore it is also possible to advect the line segments when the velocity field is given since for any node $\vec{r} = \vec{r} + \vec{u} dt$. After the advection the shortest distance is computed. In the case of no advection the sign of the distance is inherited from the original level-set function. However, when the level-set is also convected the sign must be deduced from the geometric information as well. In the current implementation each line segment is given a flag telling on which side of the element the fluid of interest is located. This directional information is then used in giving the correct sign for the distance.

The volume of the fluid of interest in the level-set method may be computed over an integral that obtains a value one inside the fluid and value zero outside the fluid. The Heaviside function $H(\phi)$ has this desired property. However, as the interface does not follow the element division the numerical integration would result into spurious fluctuations depending on the position of the interface within the elements. To obtain a smooth behavior the Heaviside function must be regularized.

$$H_\alpha(x) = \begin{cases} 0, & x < -\alpha \\ f(\alpha/x) & |x| \leq \alpha \\ 1, & x > \alpha, \end{cases} \quad (29.2)$$

where the followin has been implemented

$$f(t) = \frac{1}{2} \left(1 + \sin \left(t \frac{\pi}{2} \right) \right) \quad (29.3)$$

while one could also use

$$f(t) = \frac{3}{4} \left(t - t^3/3 \right) + \frac{1}{2}. \quad (29.4)$$

Here α is the interface bandwidth which equals typically the size of a few elements. Now the volume (area in 2D) is obtained by the integral

$$V = \int_{\Omega} H_\alpha(\phi) d\Omega. \quad (29.5)$$

After the same regularization the area (length in 2D) may be obtained from the integral

$$A = \int_{\Omega} \delta_\alpha(\phi) |\nabla \phi| d\Omega \quad (29.6)$$

where the delta function is

$$\delta_\alpha(x) = \begin{cases} 0, & |x| > \alpha \\ \frac{1}{2\alpha} \cos \left(\frac{x}{\alpha} \pi \right), & |x| \leq \alpha. \end{cases} \quad (29.7)$$

The information obtained by the above integrals may be used to improve the volume conservation of the level-set advection. If the initial volume V_0 is known the level-set function may be given a small correction by

$$d\phi = \frac{V_0 - V}{A}. \quad (29.8)$$

This correction has no physical basis but it may be argued that a consistently small update of the level-set function has a minor effect in overall results. It is more important that the volume is conserved since the history information of the shape of a bubble is gradually lost while the errors in volume are never forgotten. However, if the fluid of interest is divided into several parts this kind of overall correction does not have any justification since it could ruin the volume balance between the different domains.

The problems in accuracy may be partially resolved by using an optimal timestepping strategy. This may be achieved by looking at the velocity field around the active boundary. The normal velocity may be obtained by $u_n = \vec{u} \cdot \nabla \tilde{\phi}$. Registering the maximum velocity at band the timestep may be limited so that the Courant number is bound. If ds is the maximum allowed change in the position of the zero level-set the corresponding time-step is $dt = ds / \max |u_n|$.

In the Eulerian approach to the free surface problems the surface tension force must be smeared out to a volume force within a narrow band from the interface. The transformation is achieved by using a regularized delta function,

$$\int_{\Gamma} \sigma \kappa d\Gamma = \int_{\Omega} \sigma \kappa \delta(\phi) \nabla \phi d\Omega, \quad (29.9)$$

where σ is the surface tension coefficient and κ the curvature of the interface given by

$$\kappa = \nabla \cdot \frac{\nabla \phi}{|\nabla \phi|}. \quad (29.10)$$

In the finite element approach the force cannot be estimated directly since it involves three derivatives of the level-set function. Therefore we must solve an additional equation for the curvature κ ,

$$\kappa - c_{\kappa} \nabla^2 \kappa = \nabla \cdot \nabla \tilde{\phi}. \quad (29.11)$$

Here c_{κ} is an ad-hoc diffusion coefficient that may be used to smooth the resulting curvature field. Otherwise the sharp corners may result to very large peak values of the curvature. The weak formulation of the above equation introduces surface fluxes which are evaluated from the normal derivatives of the level-set function. Once the level-set function and the corresponding curvature have been computed the surface tension may be applied as a volume force in the flow equations.

29.3 Keywords

LevelSetSolver

This subroutine uses the finite element method to solve the equation (29.1). The implementation is valid in 2D, 3D and axisymmetric problems.

Solver solver id

Equation String "Level Set Solver"

Procedure File "LevelSet" "LevelSetSolver"

The subroutine for advecting the level-set function.

Variable String "Surface"

The name of the level-set function. This may be chosen freely as long as it is used consistently elsewhere.

Stabilize Logical

Either stabilization or bubbles are used to solve the convection problem. This flag enforces the stabilization on.

Material mat id

LevelSet Velocity i Real

The velocity field that advects the level-set function. In 2D $i=1, 2$ and in 3D $i=1, 2, 3$. This may be a constant field or also something computed with the Navier-Stokes solver.

Body Force bodyforce id

LevelSet Flux Real

The flux (i.e. the normal velocity) of the level-set function.

LevelSetDistance

This solver uses the geometric information to compute the signed distance and, if desired, to advect the zero level-set at the same time. This solver does not solve an equation and hence it does not need to have a variable of its own. The solver is limited to 2D and axisymmetric cases.

Solver `solver id`

Equation `String "Level Set Distance"`

Procedure `File "LevelSet" "LevelSetDistance"`

The subroutine for renormalizing (and advecting) the level-set function.

LevelSet Variable `String "Surface"`

This keyword should refer to the name of the level-set variable that is used to advect the field. The default is `Surface`.

Exported Variable 1 `String "Surface"`

In case the level-set variable does not exist it must be introduced. This may be the case if this subroutine is also used for advecting the level-set function.

LevelSet Convect `Logical`

Whether to also convect the level-set function. Default is `False`.

Extract Interval `Integer`

When this function is used to extract the zero level-set function the user may choose the interval how often this is done. The default is one. Just extracting the level-set may be useful if one just wants to save the zero level-set without activating reinitialization.

Reinitialize Interval `Integer`

When this function is used to reinitialize the level-set function the user may choose the interval how often this is done. The default is one but often this results to excessive smoothening of the level-set field. If reinitialization is asked the zero level-set will also be automatically extracted.

Reinitialize Passive `Logical`

If this keyword is set `True` the reinitialization is not applied to the level-set field. The field is only used to extract the zero level-set and compute the corresponding signed distance but this information is not used to change the original field.

Narrow Band `Real`

In case that also the convecting is done by this solver there is the possibility to introduce a narrow band which gives the distance at within the level-set function is recomputed. Default is ∞ . Typically this should be larger than the level-set bandwidth α used to evaluate surface integrals.

Filename `File`

The zero level-set may also be saved. It consists of a number of line segments that are defined elementwise. The results from the file may be used for visualization, for example, in MatLab. If no filename is given the zero level-set is not saved.

File Append `Logical`

If the above is given this flag enforces the results to be appended on the same file rather than writing over the old results.

Material `mat id`

LevelSet Velocity 1 `Real`

LevelSet Velocity 2 `Real`

If also convection is accounted in this solver the convection field is given by the above expressions. Currently it is not possible to give the desired surface flux as it is not uniquely defined for the line segments having different normals even at the same point.

LevelSetIntegrate

This subroutine computes the integrals (29.5) and (29.6). In addition of computing volume and surface integrals this subroutine may also be used to set the absolute level of the level-set function so that volume is conserved using equation (29.8). The implementation is valid in 2D, 3D and axisymmetric problems.

Solver solver id

Equation String Level Set Integrate

Procedure File "LevelSet" "LevelSetIntegrate"

The subroutine for computing the integrals.

LevelSet Variable String "Surface"

This keyword gives the name of the level-set function used for computing the integrals. The default is Surface.

LevelSet Bandwidth Real

When computing the values over the domain the interface is treated with smooth functions. How smooth the functions are depends on the value of this keyword. Typically the bandwidth should be such that the interface is extended over a few elements.

Conserve Volume Logical

The volume in the level-set formulation is not conserved by construction. To that end the level of the level-set function may be tuned so that conservation is enforced. The default is False.

Conserve Volume Relaxation Real

If conservation is enforced it may be done only partially as there are inaccuracies in the evaluation of the volume integrals. The default is one.

Initial Volume Real

If conservation is enforced the target volume is given by this keyword. Otherwise the volume from the first timestep is used as the target value.

LevelSetCurvature

This solver computes the value of the curvature given the level-set function using equation (29.11).

Solver solver id

Equation String Level Set Curvature

Procedure File "LevelSet" "LevelSetCurvature"

The subroutine for computing the curvature.

Variable String "Curvature"

The name of the curvature variable.

LevelSet Variable String "Surface"

This keyword gives the name of the level-set function used for computing the integrals. The default is Surface.

Curvature Diffusion Real

Artificial diffusion may be used to control the singularities of the curvature field around sharp corners. The default is zero.

Curvature Coefficient Real

A constant that is used to multiply the curvature field before the solver is exited. This may be used for example to change the sign of the curvature if the material of interest is on the outside and not on the inside.

LevelSet Bandwidth Real

The delta function for the volume force may be applied to the curvature field also within this solver directly. This has the disadvantage that the evaluation is done at nodal points rather than

at the integration points. However, if the flow solver used may not be modified this may be the best alternative. If this keyword does not exist, no delta function is used to filter the curvature field.

Boundary Condition `bc id`

Levelset Curvature BC `Logical`

The weak formulation of the curvature computation results to boundary integrals that should be set at all surfaces where the curvature is computed.

LevelSetTimestep

The solution of the level-set function is accurate only if the timestep is limited so that the local Courant number along the zero level-set is in the order of one or smaller. A tailored function for setting the timestep is given in this module. This solver assumes that the level-set variable is named `Surface` and that this variable is related to some solver. The velocity needed for setting the timestep should be given by the keywords `LevelSet Velocity i`, where $i=1, 2, 3$.

Simulation

The function call and the needed parameters reside in the `Simulation` block of the command file.

Timestep Function

`Real Procedure "LevelSet" "LevelSetTimestep"`

LevelSet Courant Number `Real`

This keyword gives the desired Courant number of for the level-set solvers. The default for the desired Courant number is one.

LevelSet Timestep Directional `Logical`

If the timestep limit is active this option may be used to account only the normal direction of the interface velocity rather than the absolute direction. Default is `False`.

Other solvers

Basically the user may give user defined material parameters where the values are computed as a function of the levelset function. Unfortunately this approach generally uses nodal points for the smearing whereas it is optimal to use the Gaussian integration points for doing this. There is one exception to this model that has been implemented for the `MaterialModels` module, namely the viscosity may be computed at Gaussian integration points.

Material `mat id`

Viscosity Model `String levelset`

This uses the levelset methodology to smear out the viscosity between inside and outside values.

Viscosity `Real`

The value of the viscosity outside the domain (negative levelset function values).

Viscosity Difference `Real`

The difference between the inside and outside viscosity values.

Levelset bandwidth `Real`

The bandwidth at which the viscosity is smeared out between the extreme values.

Model 30

Kinematic Free Surface Equation with Limiters

Module name: FreeSurfaceSolver

Module subroutines: FreeSurfaceSolver

Module authors: Thomas Zwinger, Peter Råback, Juha Ruokolainen, Mikko Lyly

Document authors: Thomas Zwinger

Document edited: November 12th 2010

30.1 Introduction

Flows with a free surface are to be found in geophysical as well as technical applications. On large scale flows the free surface usually is governed by a kinematic boundary condition given as a partial differential equation. This equation then is solved on the specific boundary in combination with the (Navier)-Stokes equation and the mesh update solver.

30.2 Theory

The implicit equation describing the free surface is given by

$$F(\vec{x}, t) = z - h(x, y, t), \quad (30.1)$$

with the explicit position of the free surface $h(x, y, t)$. Mass conservation implies that, with respect to the velocity of the surface, \vec{u}_m , F has to define a substantial surface, i.e.,

$$\frac{\partial F}{\partial t} + \vec{u}_m \nabla F = 0. \quad (30.2)$$

The net volume flux through the free surface then is given by the projection of the difference between the fluid velocity at the free surface, \vec{u} and the velocity of the free surface with respect to the surface normal

$$a_{\perp} = (\vec{u}_m - \vec{u}) \cdot \vec{n}. \quad (30.3)$$

In Geophysical context (e.g., Glaciology), a_{\perp} often is referred to as the net accumulation. With the surface unit normal defined as

$$\vec{n} = \frac{\nabla F}{\|\nabla F\|}, \quad (30.4)$$

this leads to

$$\frac{\partial F}{\partial t} + \vec{u} \nabla F = -\|\nabla F\| a_{\perp}. \quad (30.5)$$

Using the definition in (30.1), (30.5) can be rewritten in its explicit form

$$\frac{\partial h}{\partial t} + u \frac{\partial h}{\partial x} + v \frac{\partial h}{\partial y} - w = \left[1 + \left(\frac{\partial h}{\partial x} \right)^2 + \left(\frac{\partial h}{\partial y} \right)^2 \right]^{1/2} a_{\perp}, \quad (30.6)$$

with the components of fluid velocity vector at the free surface given as $\vec{u} = (u, v, w)^T$. The variational formulation of (30.6) reads as

$$\int_{\Omega} \left(\frac{\partial h}{\partial t} + u \frac{\partial h}{\partial x} + v \frac{\partial h}{\partial y} \right) \varphi dV = \int_{\Omega} \left\{ w + \left[1 + \left(\frac{\partial h}{\partial x} \right)^2 + \left(\frac{\partial h}{\partial y} \right)^2 \right]^{1/2} a_{\perp} \right\} \varphi dV, \quad (30.7)$$

where the occurrence of h in the right hand side is inserted from the previous time-step/non-linear iteration, hence linearizing the equation. In case of a horizontally moving mesh, the contribution in form of an arbitrary Lagrangian-Eulerian (ALE) formulation has to be included (by default is omitted). With the horizontal mesh velocity components, u_{mesh} and v_{mesh} , the ALE version of equation (30.6) then reads

$$\frac{\partial h}{\partial t} + (u - u_{\text{mesh}}) \frac{\partial h}{\partial x} + (v - v_{\text{mesh}}) \frac{\partial h}{\partial y} - w = \left[1 + \left(\frac{\partial h}{\partial x} \right)^2 + \left(\frac{\partial h}{\partial y} \right)^2 \right]^{1/2} a_{\perp}, \quad (30.8)$$

30.2.1 Limiters

In certain cases the free surface is constrained by an upper $h_{\text{max}}(x, y, t)$ and/or a lower $h_{\text{min}}(x, y, t)$ limit. For instance, the free surface of a fluid contained in a vessel cannot penetrate the vessel's walls. This adds the constraint

$$h_{\text{min}} \leq h \leq h_{\text{max}} \quad (30.9)$$

to (30.7) converting the variational formulation into a variational inequality. In order to obtain a consistent solution a method using Dirichlet constraints within the domain is applied. The exact procedure is the following:

1. construct the linear system: $Ah = f$, with the system matrix A and the solution vector h on the left-hand side and the force vector f on the right hand side
2. set nodes as *active* if (30.9) is violated
3. for *active* nodes the matrix and force vector are manipulated such that effectively a Dirichlet condition $h = h_{\text{max/min}}$ is applied
4. the manipulated system is solved: $\tilde{A}\tilde{h} = \tilde{f}$
5. a residual is obtained from the un-manipulated system: $R = A\tilde{h} - f$
6. an *active* node is reset if the residual is $R < 0$ (for lower limit) and $R > 0$ (for upper limit)

The whole algorithm is iterated (within the non-linear iteration loop) until the limit given in `Nonlinear System Convergence Tolerance` is reached. In the converged solution the residual represents the needed accumulation/volume flux (on matrix level, hence not in physical units) needed in order to obtain the limited solution. Consequently, the system not necessarily is volume conserving if the Dirichlet method is applied. As the solver in principle works with second order elements, the limitation procedure only converges with only the between elements shared nodes being subject to the algorithm described in this section. This is done automatically by the code.

30.3 Constraints

The code only works in Cartesian coordinates and – by the nature of the differential equation – effectively converges only in a transient simulation. Although, technically, it also can be run in steady state simulations.

30.4 Keywords

Solver solver id

Equation String "Free Surface Limited"

Variable String Varname

The change in the free surface coordinate. This may be of any name as far as it is used consistently also elsewhere, as Varname is used as a preceding keyword for the exported variable of the residual, as well as for the accumulation

Variable DOFs Integer 1

Degrees of freedom for the free surface coordinate.

Procedure File "FreeSurfaceSolver" "FreeSurfaceSolver"

The following four keywords are used for output control.

Velocity Implicitness Real

Determines the level of implicitness in the velocity field. Values shall be in the interval $c_v \in [0, 1]$. The velocity is interpolated between the current and the previous time level such that $u = (1 - c_v) u^{n-1} + c_v u^n$. Thus, unity corresponds to complete implicitness (default).

Maximum Displacement Real

This limits the maximal local displacement in a time-step. If exceeded, relaxation automatically is applied in order to limit the displacement.

Apply Dirichlet Logical

Takes the variational inequality method (here referred to as Dirichlet method) into use. The user should be aware that if the method is applied (value True) this implies setting the Nonlinear Max Iterations to a value large enough for the method to converge. The default value is False.

ALE Formulation Logical

If set to True, the mesh horizontal mesh velocity is taken into account in the convection term. The default value is False.

Relaxation Factor Real

The changes in the free surface may be relaxed. The default is no relaxation or value 1.0

Stabilization Method String

Sets stabilization method. Either Stabilized or Bubbles can be set.

Nonlinear System Convergence Tolerance Real

This keyword gives a criterion to terminate the nonlinear iteration after the maximum change in the free surface coordinate is small enough

$$\max ||dR/(R - R_0)|| < \epsilon$$

where ϵ is the value given with this keyword.

Exported Variable 1 String

The residual, which is the essential property in solving the variational inequality has to be given as an exported variable. The name is fixed by the variable name Varname given in the Solver section plus Residual. For instance, if the variable is named FreeSurf, the exported variable is expected to be FreeSurf Residual.

Exported Variable 1 DOFs Integer

As the free surface is a scalar, the value has to be set to 1.

Use Linear Elements Logical

If set to true, forces usage of linear element types despite the order of the mesh. Mind, that in case of limited elements, by default linear elements are used. The default value is False.

Equation eq id

Convection String

The type of convection to be used: None (default), Computed, Constant. In the last case, the keyword Convection Velocity is expected to be found in the Material section.

Body Force bf id

Varname Accumulation Real

sets the value for the normal accumulation/volume flux, a_{\perp} for the variable name varname. If this keyword is set, the following keyword Varname Accumulation Flux is ignored (as those are excluding)

Varname Accumulation Flux i Real

sets the accumulation flux in Cartesian components ($i = 1,2,3$ in 3-dimensional problem). The resulting vertical flux then is evaluated using the surface normal.

Initial Condition ic id

Varname Real

Initiation of the free surface variable (sets initial shape of surface)

Boundary Condition bc id

Body ID Integer

usually, the solver is run on a lower dimensional boundary of the model. Then a separate body-id has to be defined and all component of the solver (Equation, Body Force, Equation, Initial Condition and Material) defined accordingly.

Varname Real

Dirichlet condition of the free surface variable (makes really sense only on dimension - 2 boundaries, e.g. lines in case of a three dimensional run)

Mesh Update i Real

usually, the free surface evolution should have a feedback on the domain's geometry. This usually is achieved by running the MeshUpdate Solver and linking the variable of the free surface with the corresponding component of the Mesh Update ($i=1,2,3$). For instance, in a 3-dimensional case with the variable name FreeSurf this could read as: Mesh Update 3 = Equals FreeSurf

Model 31

Free Surface with Constant Flux

Module name: FreeSurfaceReduced

Module subroutines: FreeSurfaceReduced

Module authors: Peter Råback

Document authors: Peter Råback

Document edited: August 5th 2002

31.1 Introduction

The determination of free surface is often an essential part of solving a fluid dynamics problem. Usually the surface is found by solving a free surface equation resulting from force balance, or by finding the free surface from zero flux condition. In some extreme cases both of these methods were found to fail and therefore an alternative approach was taken. The method can only be applied to stationary 2D or axisymmetric flows where the total flux is conserved. This is the case, for example, in many coating and drawing processes.

31.2 Theory

The determination of the free surface takes use of the conservation of mass. If the flow is stationary the mass flux through all planes cutting the flow must be same. In the following we concentrate on the axisymmetric case which has more applications than the 2D case.

In the axisymmetric case the mass flux is obtained from

$$f(R, z) = \int_{R_0}^R (\vec{u} \cdot \vec{n}) r ds. \quad (31.1)$$

The free surface is set by finding a surface profile $R(z)$ such that the integral is constant for all nodes on the surface, or

$$f(R, z_j) = f(R_1, z_1) \quad \forall j \in [1, M]. \quad (31.2)$$

Note that the factor 2π has been consistently omitted since it has no bearing to the shape of the free surface.

The subroutine uses simple heuristics to determine the direction of the flow on the free surface. The first upwind node z_1 on the free surface is assumed to be fixed and the corresponding flux is f_1 . The new radius is set approximately by assuming that the added or removed flow has the same velocity as the velocity on the surface. Then the corrected radius is found from

$$u_n R^{(m)} dR^m = f(R^{(m)}, z) - f(R_1, z_1) \quad (31.3)$$

or

$$R^{(m+1)} = R^{(m)} + \frac{f(R^{(m)}, z) - f(R_1, z_1)}{u_n R^{(m)}}. \quad (31.4)$$

After the new profile is being found the element nodes are moved to the new positions. The nodes that are not on the surface may be mapped in many different ways. The straight-forward strategy is to use linear 1D mapping. Also more generic 2D mapping may be used.

The free surface and the fluid flow must be consistent and therefore the system must be solved iteratively. When convergence of the coupled system has been obtained the suggested dR vanishes and the free surface solver does not affect the solution.

Sometimes the free surface solver overshoots and therefore it may be necessary to use relaxation to suppress the large changes of the solution.

Note that the free surface solver is simple based on mass conservation. No forces are applied on the free surface. If surface tension needs to be taken into account it may be done while solving the Navier-Stoke equation.

31.3 Applicable cases and limitations

The method has some limitations which are inherent of the method:

- Limited to steady-state simulations.
- Limited to 2D and axisymmetric cases.
- If there is back-flow within the free surface flow the correctness of the solution is not guaranteed.

Some limitations result from the current implementation:

- The free surface must be oriented so that the flow is on its negative side.
- There may be several free surfaces of this type but they must be directed the same way.
- The line integral from R_0 to R may cause some difficulties in unstructured meshes. Therefore structured meshes are favored.
- At the moment density is assumed to be constant and therefore only incompressible fluids may be considered.

31.4 Keywords

Solver `solver id`

Equation `String "Free Surface Reduced"`

Variable `String dx`

The change in the free surface coordinate. This may be of any name as far as it is used consistently also elsewhere.

Variable DOFs `Integer 1`

Degrees of freedom for the free surface coordinate.

Procedure `File "FreeSurfaceReduced" "FreeSurfaceReduced"`

The following four keywords are used for output control.

Perform Mapping `Logical`

If this keyword is `True` the coordinate mapping is done locally by using linear 1D mapping. This is also the default. Also 2D mapping is possible by using a separate mesh update solver. Then the keyword should be set to `False`.

Nonlinear System Relaxation Factor `Real`

The changes in the free surface may be relaxed. The default is no relaxation or value 1.0

Nonlinear System Convergence Tolerance Real

This keyword gives a criterion to terminate the nonlinear iteration after the maximum change in the free surface coordinate is small enough

$$\max ||dR/(R - R_0)|| < \epsilon$$

where ϵ is the value given with this keyword.

Boundary Condition bc id

Free Surface Reduced Logical

Must be set to True for the free surface when the solver is used. The boundary must be simply continuous.

Free Surface Number Integer

If more than one free surface of the reduced type is present simultaneously they must somehow be separated. This keyword is for that purpose. The surfaces should be ordered from 1 to the number of free surfaces. Value 1 is also the default if the surface is active. Note that free surfaces with different numbers should be aligned the same way and should not touch each other.

Free Surface Bottom Logical

If this flag is free it sets the lower boundaries of integration when solving for the free surface. Note that this surface should not touch any of the free surfaces. A free surface is automatically a lower boundary for another free surface.

If mapping is not performed within the solver also boundary conditions for the mapping are required. Surface tension may be taken into account while solving the Navier-Stokes equation. The proper keywords for activating the surface tension are explained in the manual of the Navier-Stokes solver.

Model 32

Transient Phase Change Solver

Module name: PhaseChangeSolve

Module subroutines: TransientPhaseChange

Module authors: Peter Råback

Document authors: Peter Råback

Document created: 22.10.2004

Document edited: 14.12.2009

32.1 Introduction

There are many phenomena that involve an interface between liquid and solid phase. Such problems occur, for example, in crystal growth and casting processes. This subroutine defines the position of the phase change boundary in a transient case using an Lagrangian approach.

For Lagrangian steady state phase change algorithm look at the next chapter. For Eulerian phase change algorithm look at the enthalpy method in the heat solver. Generally Lagrangian approaches are more accurate but their use is limited to rather smooth interfaces with moderate displacements.

32.2 Theory

General theory

The phase change from solid to liquid occurs at the melting point T_m . At the boundary the temperatures of the liquid and solid are therefore equal to that. The phase change results to a change in the internal energy known as the latent heat L .

The latent heat makes the diffusive heat flux over the boundary discontinuous and results to the so called Stefan condition

$$L\rho\vec{v} \cdot \vec{n} = (\kappa_s \nabla T_s - \kappa_l \nabla T_l) \cdot \vec{n}, \quad (32.1)$$

where \vec{n} is the normal of the phase change boundary, \vec{v} is the velocity of the phase change boundary, ρ is the density of the solid and T_s and T_l are the temperatures of the solid and liquid phases, and κ_s and κ_l are the thermal conductivities, respectively.

In steady state pulling and drawing processes the velocity of the phase change boundary should be equal to pull velocity, $\vec{v} = \vec{V}$ (bulk velocity of the solid phase).

Transient algorithm

In transient phase change problems the interface temperature is set to be at the melting point when solving the heat equation. From the solution a heat flux is then obtained from

$$\vec{q} = \kappa_s \nabla T_s - \kappa_l \nabla T_l. \quad (32.2)$$

Now this heat flux is assumed to be used for the melting of the solid phase into liquid phase. Assuming that the phase change boundary is mapped to the new position moving it only in the y -direction we get from equation (32.1) the velocity in the y -direction,

$$\rho L n_y (v_y - D_v \nabla^2 v_y) = \vec{q} \cdot \vec{n}. \quad (32.3)$$

Here an artificial diffusion D_v has been added since the algorithm otherwise is prone to numerical oscillations. In order for the diffusion not to affect the results significantly it must fulfil the condition $D_v \ll h^2$ where h is the size of the 1D elements.

The corresponding displacement is easily obtained from multiplication $u_y = v_y dt$, where dt is the timestep. However, in the current formulation may also be done using the Galerkin method to include the possibility of an additional diffusion factor. Therefore the equation is of the form,

$$\frac{\partial u_y}{\partial t} - D_u \nabla^2 u_y = v_y. \quad (32.4)$$

In continuous processes the triple point may be used to define the pull velocity so that at the point the solution of the equation vanishes. In case the pull occurs in the y -direction this means that $V_y = v_y$.

The algorithm is ideally suited for relatively small time-steps where the change in the position is small compared to the other dimensions of the problem. Otherwise the transient algorithm may result to spurious oscillations. However, often the timestep size is most severely limited by the flow computations. Therefore it may be possible to boost the convergence towards the true operation regime by multiplying the suggested change by a constant factor.

32.3 Applicable cases and limitations

The method has some limitations which are described below

- Phase change surface must be nearly aligned with either of the main axis. To be more precise the boundary must in all instances be such that for each coordinate there is only one point on the boundary.
- Applicable in 2D and 3D cases
- Melting point and density over the interface may vary is assumed to be constant over the whole interface.
- The convection velocity of the interface should be constant.
- It should be noted that the solver only gives the position of the phase change boundary. In order to modify the whole geometry a mesh update solver must be applied.

32.4 Keywords

Solver `solver id`

Equation `String` "Transient Phase Change"

Procedure `File` "TransientPhaseChange" "TransientPhaseChange"

The subroutine that performs the phase change analysis.

Variable `String` PhaseSurface

The variable for the PhaseSurface coordinate. This may be of any name as far as it is used consistently also elsewhere.

Variable DOFs `Integer` 1

Degrees of freedom for the free surface coordinate, the default.

Phase Change Variable `String`

By default the phase change analysis uses Temperature as the active variable. The analysis may be performed also to any other scalar variable given by this keyword

Use Nodal Loads Logical

The most accurate method of computing the heat fluxes is to use the residual of the matrix equation. This is activated by the keyword `Calculate Loads` in the heat equation and it results to a variable named `Temperature Loads` that may be used directly to give the melting heat over the interface nodes.

Normal Variable String

The normal of an element may be computed directly from each element segment, or it may be computed using Galerkin method in the `NormalSolver`. In the latter case the name of the normal field variable may be given by this keyword.

Triple Point Fixed Logical

This keyword enforces the triple point to be fixed. Depending on the type of algorithm this may mean different things. In the transient algorithm this means that the interface velocity is tuned so that the velocity at the triple point is zero. Only applicable in 2D where the triple point is unique.

Pull Rate Control Logical

The pull rate may be set so that the triple point remains at a fixed position. The feature is activated setting this keyword `True`.

Velocity Relaxation Factor Real

The relaxation factor for the interface velocity field.

Velocity Smoothing Factor Real

The velocity diffusion factor of the interface, D_v .

Transient Speedup Real

The factor at which the change in the boundary position is changed in the transient case. This may be used to speedup the transient convergence.

Nonlinear System Max Iterations Integer

In case the pull-rate control is used the phase change algorithm may have to be solved several times in order to define the consistent pull-rate. This keyword gives the maximum number of iterations.

Nonlinear System Convergence Tolerance Real

The tolerance for terminating the transient algorithm.

Body body id

Solid Logical

Liquid Logical

The solver requires information on which of the materials in the system is solid and which is liquid. Currently the solver assumes that both the liquid and solid is uniquely defined.

Material mat id

Heat Conductivity Real

In a transient case the heat conductivities of the both materials must be given.

Density Real

Density is needed to obtain the latent heat in units of energy per volume.

Latent Heat Real

The latent heat is the specific internal energy related to the phase change. The latent heat may also be a variable.

Convection Velocity i Real

For the transient algorithm the pull velocity of the boundary may be given with this keyword.

Boundary Condition bc id

Body Id Integer

The phase change solver operates usually on a boundary of a two-dimensional domain. Technically the equation on the boundary is treated in a normal finite element manner and therefore the boundary must be defined to be the body where the equation is to be solved. Usually this would be the next free integer in the list of bodies.

Phase Change Side Logical

This keyword is used for the boundaries that define the edges of the phase change interface. The diffusive operators used for smoothing create a weak term in the Galerkin formulation that must be cancelled. When this flag is active the weak terms are not assembled at all for the boundary thus eliminating the need to cancel them.

Model 33

Steady State Phase Change Solver

Module name: PhaseChangeSolve

Module subroutines: SteadyPhaseChange

Module authors: Peter Råback and Juha Ruokolainen

Document authors: Peter Råback

Document created: 22.10.2004

Document edited: 14.12.2009

33.1 Introduction

There are many phenomena that involve an interface between liquid and solid phase. Such problems occur, for example, in crystal growth and casting processes. This subroutine defines the position of the phase change boundary by finding the correct isotherm in a steady state simulation. The mesh is the correspondingly mapped i.e. this is a Lagrangian approach.

For transient phase change algorithms look at the next chapter. For Eulerian phase change algorithm no additional solver is required as the phase change is implemented within the heat equation of Elmer by using the enthalpy method. Generally Lagrangian approaches are more accurate but their use is limited to rather smooth interfaces with moderate displacements.

33.2 Theory

For the general theory on phase change look at the Theory section of the previous chapter.

Steady state algorithm

In steady state the algorithm is based mainly on geometrical ideas. First the heat equation for temperature T is solved by using a flux condition for the interface

$$q = L\rho\vec{V} \cdot \vec{n}. \quad (33.1)$$

Thereafter the next approximation for the phase change surface may be found by going through each element and creating a list of line segments E_j on the isosurface. This is basically the zero level-set of the field $T - T_m$. Each line segment is defined by two coordinate $\vec{x}_{j,1}$ and $\vec{x}_{j,2}$. The surface is then updated by mapping the current phase change surface to the line segments. For the moment a N^2 algorithm is used for the mapping. For larger cases a more robust search algorithm might be implemented.

For example, if a free surface is almost aligned along the x-axis, then for a node (x_i, y_i) on the boundary the proposed change of the point i in the y-direction is

$$s_y = (y_{j,1} - y_i) + (x_i - x_{j,1}) \frac{y_{j,2} - y_{i,1}}{x_{j,2} - x_{j,1}} \quad (33.2)$$

assuming that $x_i \in [x_{j,1}, x_{j,2}]$ while $s_x = 0$.

Speeding up the convergence

In many cases the simple geometrical search algorithm converges very slowly. The reason is the explicit character of the algorithm that fails to account for the change in the temperature field caused by the moving phase change boundary. This limitation may be partially overcome using suitable under- or over-relaxation. This relaxation parameter may also be tuned during the iteration using lumped quantities such as the proposed change in the volume of the phases that may be expressed as

$$U = \int_A \vec{s} \cdot \vec{n} dA. \quad (33.3)$$

The proposed volume changes form a series, $U^{(0)}, U^{(1)}, \dots, U^{(m-1)}, U^{(m)}$. Assuming that the series is a geometric one we may estimate the required relaxation factor that would give the correct phase change boundary at just one iteration,

$$c^{(m)} = c^{(m-1)} \frac{U^{(m-1)}}{U^{(m-1)} - U^{(m)}}. \quad (33.4)$$

In numerical tests this formula was found occasionally to overshoot and therefore a less aggressive version is used instead,

$$c^{(m)} = c^{(m-1)} \frac{1}{2} \frac{U^{(m-1)} + U^{(m)}}{U^{(m-1)} - U^{(m)}}. \quad (33.5)$$

The use of the lumped model requires that the temperature field is described accurately enough. To ensure numerical stability the factor c should have an upper and lower limits. After the factor has been defined the suggested displacements are simply scaled with it, $\vec{s}' = c\vec{s}$.

It is also possible to accelerate the solution locally using a Newton kind of iteration. If the basic algorithm has already been applied at least twice we may estimate the sensitivity of the local temperature to the moving interface and using this information to estimate a new change,

$$s^{(m)} = \frac{T_m - T^{(m)}}{T^{(m)} - T^{(m-1)}} s^{(m-1)}. \quad (33.6)$$

This algorithm might be a better option if the phase change surface is such that there is not much correlation between the displacements at the extreme ends. However, the algorithm may be singular if the isotherms of consecutive iterations cross. Any point i where $T_i^{(m-1)} \approx T^{(m)}$ leads to problems that may be difficult to manage. This handicap may rarely limit the usability of the otherwise robust and effective scheme.

33.3 Applicable cases and limitations

The method has some limitations which are described below

- Limited to steady state cases
- Limited to 2D and axisymmetric cases.
- Phase change surface must be nearly aligned with either of the main axis. To be more precise the boundary must in all instances be such that for each coordinate there is only one point on the boundary.
- Melting point is assumed to be constant over boundary (not concentration dependent, for example).
- It should be noted that the solver only gives the position of the phase change boundary.
- When internal mesh update is used the mesh is distorted only in one coordinate direction.

33.4 Keywords

Solver `solver id`

It is worth noting that for this solver the problem is, or at least could be, solved accurately. All the nonlinearities of the problem reside in the coupling with the heat equation. Hence, there is no point in giving criteria on the nonlinear system level. Only the coupled level tells whether the system has truly converged.

Equation `String "Steady Phase Change"`

Procedure `File "SteadyPhaseChange" "SteadyPhaseChange"`

The subroutine that performs the phase change analysis.

Variable `String Surface`

The variable for the PhaseSurface coordinate. This may be of any name as far as it is used consistently also elsewhere.

Variable DOFs `Integer 1`

Degrees of freedom for the free surface coordinate, the default.

Phase Change Variable `String`

By default the phase change analysis uses `Temperature` as the active variable. The analysis may be performed also to any other scalar variable given by this keyword

Nonlinear System Relaxation Factor `Real`

Giving this keyword triggers the use of relaxation in the phase change solver. Using a factor below unity may sometimes be required to achieve convergence. Relaxed phase change variable is defined as follows:

$$u'_i = u_i + \lambda s_{i-1},$$

where λ is the factor given with this keyword. The default value for the relaxation factor is unity. If using the lumped model to accelerate the solution the final relaxation factor will be the product of the two.

Nonlinear System Newton After Iterations `Integer`

The local Newton type of iteration may be set active after a number of iterations given by this keyword.

Nonlinear System Newton After Tolerance `Real`

The Newton type of iteration may also be activated after a sufficiently small change in the norm. This keyword gives the limit after which Newton iteration is triggered on.

Lumped Acceleration After Iterations `Integer`

The phase change solver may be accelerated pointwise, or by using a lumped model to determine an optimal relaxation factor for the whole solution. This keyword activates the lumped model procedure.

Lumped Acceleration Mode `Integer`

This helps to toggle between different versions of the lumped acceleration. The options include values 0,1,2,3 where 0 is also the default.

Lumped Acceleration Limit `Real`

The lumped approach sometimes gives too high or too small relaxation factors. This may happen particularly at the very vicinity of the solution where the approximation errors have a greater effect.

Triple Point Fixed `Logical`

This keyword enforces the triple point to be fixed. This means that the temperature used for finding the isotherm is set to be the temperature of the triple point. This freezes the position by construction. Typically this should be combined with a temperature control that at convergence results to the triple point being at melting point.

Internal Mesh Movement Logical

The mesh around the growth interface may be moved in two ways: using the mesh update solver based on the linear elasticity, or using the simple 1D mapping built in the solver. If this flag is set active the internal mesh movement is used.

Passive Steps Integer

If for some reason we want to omit that the solved phase change position is updated to the mesh we may use this flag which for the given number of rounds does not apply the mesh update.

Body body id

Solid Logical

Liquid Logical

The solver requires information on which of the materials in the system is solid and which is liquid. Currently the solver assumes that both the liquid and solid is uniquely defined.

Material mat id

Melting Point Real

The melting point is the temperature at which the transition from solid to liquid occurs. The melting point is assumed to be constant. If the triple point is fixed the value of the melting point is not used in finding the levelset.

Density Real

Density may be needed in the computation of the surface normals. By default, the normals point out from the denser of the two materials. Also the density is needed for the computation of latent heat release.

Latent Heat Real

The latent heat is the specific internal energy related to the phase change. The latent heat may also be a variable. It is actually not needed by the phase change solver but must be provided for the heat solver.

Boundary Condition bc id

Phase Change Logical

The interface of the phase change is determined by this special flag.

Phase Velocity i Real

For the steady state case the heat equation often requires the heat flux as a boundary condition. For this reason the phase velocity for each component may be determined. The keyword is not needed by the current solver.

Model 34

Particle Dynamics

Module name: ParticleDynamics

Module subroutines: ParticleDynamics

Module authors: Peter Råback, Juha Ruokolainen

Module status: Alpha

Document authors: Peter Råback

Document created: 16.10.2010

Document edited: 17.10.2010

34.1 Introduction

Note: this is an initial version of the dynamic particle tracker. For real applications it probably requires some additional effort.

The ability to follow single or statistical particles within a finite element can be used in a variety of applications. A common application is to follow particles along streamlines for the purpose of flow visualization. Accounting for electrostatic forces opens the field to microfluidics and accounting for the gravitational force enables applications in sedimentation, for example. If also particle-particle interaction is accounted for also granular flow phenomena may be studied.

This module depends on the many library routines related to particle transport in Elmer. In this module it is assumed that there may be particle-particle interactions. This choice fixes the time-stepping strategies of the different particles together, at least without heroic timestepping schemes. In other words, the same timestep size is applied to the whole particle set.

The particles are located in the finite element mesh using a marching routine where intersections with element boundaries are checked for. The nearest boundary on the way is crossed until there is no boundary to cross. Then the right element has been reached. The algorithm is fast when the stepsize with respect to elementsize is smaller or of the same order. Therefore for the initialization the octree-based search may be more economical and also more robust regarding geometric shapes.

The particle-particle interaction is based on the knowledge of nearest neighbours. Currently the neighbours are determined using the closeness to the nodes of the parent element. This means that the interaction distance needs to be smaller than $h/2$ where h is the mesh parameter. Further, it means that the mesh must be rather uniform.

As the name implies, this module assumes the particles to be dynamic i.e. they have an acceleration. However, the user may also use the module neglecting the inertial forces and requiring a force balance between the drag force and external forces.

34.2 Theory

Forces acting on the particle

Assume that we have a particle in position \vec{r} . The corresponding velocity is

$$\vec{v} = \frac{d\vec{r}}{dt} \quad (34.1)$$

Newton's second law yields

$$m \frac{d\vec{v}}{dt} = \Sigma f(\vec{r}, \vec{v}, \dots) \quad (34.2)$$

where a number of different forces may be considered.

The gravity force acting on the particle is

$$\vec{f} = m\vec{g}, \quad (34.3)$$

where \vec{g} is the acceleration due to gravity. The electrostatic force is simply proportional to the electric field

$$\vec{f}_e = q\vec{E} = q\nabla\phi \quad (34.4)$$

where q is the electric charge.

The viscous fluids cause also a force that acts on the particle

$$\vec{f}_s = -b(\vec{v} - \vec{v}_0) \quad (34.5)$$

where \vec{v}_0 is the velocity of the fluid. If the change is estimated to be $d\vec{r}$ then the estimate may be improved by the gradient of velocity, i.e. $\nabla\vec{v}_0 \cdot d\vec{r}$. For Stokes flow the proportionality coefficient scales with viscosity, for example for spheres $b = 6\pi\eta d$ where η is the fluid viscosity and d the radius of the sphere.

Collision model

Two particles may collide with one-another. Assume that the initial particle positions are \vec{r}_1 and \vec{r}_2 . Velocity vectors are \vec{v}_1 and \vec{v}_2 and let's define $\delta\vec{r} = \vec{r}_1 - \vec{r}_2$ and $\delta\vec{v} = \vec{v}_1 - \vec{v}_2$. Now the condition for a collision is

$$|\delta\vec{r} + \delta\vec{v} dt| = R_1 + R_2. \quad (34.6)$$

This leads to condition for the timestep

$$dt = \frac{-b - \sqrt{b^2 - ac}}{a} \quad (34.7)$$

where $b = \delta\vec{r} \cdot \delta\vec{v}$, $a = \delta\vec{v} \cdot \delta\vec{v}$, and $c = \delta\vec{r} \cdot \delta\vec{r} - (R_1 + R_2)^2$. Collision happens if $0 < dt < Dt$.

The collision only affects the normal component. The normal vector is aligned with $\delta\vec{r}' = \delta\vec{r} + dt \delta\vec{v}$ i.e. $\vec{n}_r = \delta\vec{r}' / |\delta\vec{r}'|$. Now the normal velocity components are $v_{i,n} = \vec{v}_i \cdot \vec{n}_r$. After the collision the normal velocity component is

$$v'_{1,n} = \frac{cM_2(v_{2,n} - v_{1,n}) + M_1v_{1,n} + M_2v_{2,n}}{M_1 + M_2} \quad (34.8)$$

and likewise for $v'_{2,n}$. Here the parameter c is called bounciness and it varies between zero, for fully inelastic collision, to one, for fully elastic collisions. The new velocity is now

$$\vec{v}'_i = \vec{v}_i + (v'_{i,n} - v_{i,n})\vec{n}_r \quad (34.9)$$

and the new position,

$$\vec{r}'_i = \vec{r}_i + \vec{v}_i dt + \vec{v}'_i dt' \quad (34.10)$$

where $dt' = Dt - dt$.

Collisions with the wall are governed with the same equations assuming that mass of the wall is infinite.

The change in the velocity and coordinate position may be mutated to a change in velocity and force. This way the collision model is better additive with the other type of models present in the system.

Contact model

In general the contact between particles depends on their relative position, relative velocity, and relative angular velocity. Generally the contacts should include some damping (negative feedback from velocity) since otherwise the system is prone to flow up. In molecular dynamics, for example, also interaction with more than two particles should be considered. The current treatment is quite limited and we here assume that the contact results just to a spring force in the form

$$\vec{f}_k = k \max(R_1 + R_2 - |d\vec{r}|, 0) \vec{n}_r, \quad (34.11)$$

where k is the spring coefficient.

A similar particle contact model may be present with the wall but possibly with different value for the spring coefficient.

Periodic boundary conditions

It is relatively straight-forward to implement periodic boundary conditions for rectangular and hexahedral type of geometries. And for different geometries the periodic conditions seems more unlikely.

Time evolution

For particles with mass the basic update sequence of velocity and position is

$$\vec{v}_{i+1} = \vec{v} + \frac{dt}{m} \Sigma f \quad (34.12)$$

$$\vec{r}_{i+1} = \vec{r} + dt \vec{v}_{i+1} \quad (34.13)$$

while for massless particles it is assumed that the particle drag is in balance with the other forces given explicitly

$$\vec{v}_{i+1} = \frac{1}{b} \Sigma f \quad (34.14)$$

$$\vec{r}_{i+1} = \vec{r} + dt \vec{v}_{i+1} \quad (34.15)$$

The timestep dt may be given explicitly, or it may be defined from the characteristic velocity V . If the change in distance dS is given then

$$dt = \frac{dS}{V}, \quad (34.16)$$

and when the Courant number C is given

$$dt = C \frac{h}{V}, \quad (34.17)$$

where V is either the maximum absolute velocity, or the mean absolute velocity.

Also other timestepping schemes could be used but that's something for later.

34.2.1 Postprocessing

The possibility to use each particle as an integration point in data fitting problem makes it possible to couple the particles back to a continuous field. The following kinds of information could be abstracted from the particles, for example.

Kinetic energy of particles

$$E_k = \frac{1}{2} m v^2. \quad (34.18)$$

Potential energy associated to gravity field

$$E_g = m \vec{g} \cdot \vec{r}. \quad (34.19)$$

Potential energy associated to electrostatic field The corresponding potential energy is

$$E_e = q\phi. \quad (34.20)$$

Etc. In practice sufficient amount of data may not be present at every node if the data is used only after appropriate smoothing. !and hence some regularization may be applied to the fitting problem. !In practice this means adding some diffusion to the fitting problem.

34.3 Keywords

Solver `solver id`

Equation `String [ParticleDynamics]`
The name of the equation.

Procedure `File "ParticleDynamics" "ParticleDynamics"`
The name of the procedure.

Keywords related to the allocation and initialization of the particles.

Number of Particles `Integer`
Number of particles to be sent. The number may be given by this keyword as an absolute number. Often a relative number, particularly in parallel computation, may be favorable.

Particle Node Fraction `Real`
The relative fraction of particles to nodes. The nodes may also be masked ones.

Particle Element Fraction `Real`
The relative fraction of particles to elements. The elements may also be masked ones.

Coordinate Initialization Method `String`
Initialization method for the coordinates. The options include `nodal ordered`, `elemental ordered`, `sphere random`, `box random`, `box random cubic` with their own initialization strategy.

Initial Coordinate `Size n, dim; Real`
The default initialization methods for coordinates.

Initialization Condition Variable `String`
If this is given then the particles are initialized only where this has a nonzero permutation vector.

Initialization Mask Variable `String`
If this is given then the particles are initialized only in elements or nodes where the variable has a positive value.

Min Initial Coordinate `i Real`

Max Initial Coordinate `i Real`
For box initialization methods set the bounding box for doing initialization.

Particle Cell Radius `Real`
If the initialization method is `box random cubic` then the particle is always put to a unit cell located in the given bounding box.

Particle Cell Fraction `Real`
If the initialization method is `box random cubic` then this keyword gives the fraction of filled cells in the initial configuration.

Initial Sphere Radius `Real`
If the initialization method is `sphere random` then this set the radius of the sphere.

Initial Sphere Center `Size 3; Real`
Sets the size of the initial sphere center.

Velocity Initialization Method String

There are many ways to initialize the velocities of the particles: thermal random, even random, constant random.

Initial Velocity Size n , dim; Real

The particle velocities may be also initialized only by this keyword, or this may be used to give a bulk component to the otherwise random velocity field.

Initial Velocity Amplitude Real

In many velocity initialization methods an initial velocity amplitude is needed.

Initial Velocity Time Real

When initializing the velocity also the initial coordinates may be affected by determining a offset for the time used to advance the particles. This could be used, for example, to distribute the particles from an initial point using the random velocity field.

Initial Coordinate Search Logical

After the initialization is done do an initial octree-based search for the initial coordinate positions. This is applicable only to serial problems.

Reinitialize Particles Logical

Reinitialize the particles in the start of each time when the subroutine is called. This would make sense in some kind of scanning mode. The default is False.

Particle Release Number Integer

If not all particles are sent at the same time. This is the absolute number of particles sent at the start of the subroutine call.

Particle Release Fraction Real

If not all particles are sent at the same time. This is the fraction of particles sent at the start of the subroutine call.

Delete Wall Particles Logical

Currently a hack which is used to remove particles sitting on the wall which otherwise seem to get stuck.

Keywords related to the timestepping strategy.

Timestep Size Real

The internal timestep size.

Max Timestep Size Real

The lower limit of the internal timestep size.

Min Timestep Size Real

The upper limit of the internal timestep size.

Timestep Distance Real

The distance that is travelled within one timestep based on the characteristic velocity.

Timestep Courant Number Real

The desired courant number resulting from the timestep based on the characteristic velocity. Note that currently just one element is used to compute the parameter h . A global definition of the courant number would result to a significant increase in the computational cost.

Max Characteristic Speed Logical

When computing characteristic velocity use the max norm.

Max Timestep Intervals Integer

Maximum number of internal timesteps.

Max Cumulative Time Real

Maximum cumulative time within one call.

Simulation Timestep Sizes Logical

Alternatively, one may use the timesteps as defined by the Timestep Sizes of the Simulation section.

Keywords related to the actual physical interaction models chosen within the particles and with particles and walls.

Particle Particle Collision Logical

Is there some collisions between particles.

Particle Particle Contact Logical

Is there contact between particles resulting to additional forces.

Box Particle Periodic Logical

Is the system periodic.

Box Periodic Directions Integer

If not all directions require the periodic model this may be used to define the active directions.

Box Particle Collision Logical

Is there collisions between particles and 2D or 3D box. This provided for a cheaper treatment of BCs than the generic way.

Box Particle Contact Logical

Is there contact between particles and walls resulting to additional force.

Box Contact Directions Integer

If not all directions require the contact model this may be used to define the active directions.

Velocity Variable Name String

Name of the variable if velocity drag is present.

Velocity Gradient Correction Logical

When using the drag model evaluate the drag forces using correction from the velocity gradient.

Potential Variable Name String

Name of the variable if electrostatic potential is present.

Velocity Condition Variable Name String

Name of the field which determines the fixed velocity conditions of the particles.

Coordinate Condition Variable Name String

Name of the field which determines the fixed coordinate conditions of the particles.

Keywords related to the physical properties of the particle and to the joint physical properties of the particle-particle and particle-wall contacts.

Particle Mass Real

There are a number of particle properties needed in different interaction models and particle mass is one of them. In principle these could be altered to be variables but currently they are assumed to be the same for all particles.

Particle Radius Real

The particle radius used in particle-particle interaction, and in evaluating the density of the particle.

Particle Gravity Logical

Should gravity be accounted for. If yes, use the gravity defined in the

Particle Lift Logical

The background fluid has a density that results to a lift (bouancy) that may be accounted for. Should gravity be accounted for. If yes, use the gravity defined in the Constants section.

Particle Damping Real

Particle damping proportional to velocity only.

Particle Drag Coefficient Real
Particle drag coefficient in fluid field.

Particle Bounciness Real
Defines, when particles collide is the collision totally elastic or totally inelastic. Corresponding extreme values are 1 and 0. This relates only to collision models.

Particle Spring Real
Spring constant in the force model between particles. This relates only to contact models.

Particle Charge Real
The electric charge of the particle.

Particle Decay Distance Real
The decay of the particle effect.

Wall Particle Radius Real
In interaction with the walls different properties are given as the interaction with the wall is quite different regarding, for example, the contact shape.

Wall Particle Spring Real
Spring constant in interaction with wall.

Wall Particle Bounciness Real
Elasticity of collision with interaction with the wall.

Keywords related to the generation of fields from the particle data.

Particle To Field Logical
Is there any coupling from particles to field needed? This leads to the need of finite element machinery. To opposite is always assumed to be true i.e. the particles are always assumed to be located in the FE mesh.

Reinitialize Field Logical
When revisiting the solver should the particle field be initialized at the start.

Particle To Field Mode Integer
If a field is generated from the particles, what actually should be computed.

Particle Decay Time Real
This is an optional parameter that represents the characteristic time that is used to forget history data from the particle to field representation.

Particle Decay Distance Real
This is an optional parameter that represents the characteristic distance that is used to forget history data from the particle to field representation.

Keywords related to saving and echoing information. No effect to the actual computations.

Output Interval Integer
The internal output interval of the solver. If not given the no particle data will be saved within the solver. The alternative is to save particle data with an external solver.

Output Format String
Output format which may be either table or vtu.

Table Format Logical

Vtu Format Logical
Alternative way of giving the output format. Has the nice property that several formats may be given at the same time.

Filename Prefix String
The prefix of the filename used for saving. Depending on the chosen format an appropriate suffix is attached to the prefix.

Filename Particle Numbering Logical

If possible in the format, use particle indexes for the numbering of files.

Filename Timestep Numbering Logical

If possible in the format, use timestep indexes for the numbering of the files. This is the default in vtu format.

Particle Info Logical

Optionally print out on the screen information on the number of particles and time steps taken.

Statistical Info Logical

Optionally print out on the screen some statistical information on the coordinate positions and velocities. May be useful for debugging purposes, for example.

Scalar Field i String

The scalar fields of the particles to be saved in vtu format. Currently options include distance and dt.

Vector Field i String

The vector fields of the particles to be saved in vtu format. Currently options include velocity and force.

Particle Save Fraction Real

If there is a huge number of particles it may be sufficient to use only a subset of them for visualization. This keyword gives the fraction.

Boundary Condition bc id

Wall Particle Collision Logical

This activates the collision model between particles and generic boundaries.

Particle Accumulation Logical

An optional flag that activates the possible destruction of the particles at the boundary in case conditions for accumulation are met.

Particle Accumulation Max Speed Real

If this critical speed is given, then accumulate only those particles with smaller velocity.

Particle Accumulation Max Shear Real

If this critical shear rate is given, then accumulate only those particles with a smaller shear rate.

Particle Trace Logical

If this flag is set active then use the accumulated particles to compute a trace to a finite element field.

Moving Wall Logical

The movement of the wall may be accounted for in the wall-particle collision model.

Model 35

Semi-Lagrangian Advection Using Particle Tracking

Module name: ParticleAdvect

Module subroutines: ParticleAdvect

Module authors: Peter Råback, Juha Ruokolainen

Module status: Alpha

Document authors: Peter Råback

Document created: 16.6.2010

Document edited: 16.6.2010

35.1 Introduction

This solver utilizes the particle tracker features of Elmer to advect scalar fields diffusion-free on the mesh. For each node of the field one particle is sent backwards in time and the field value is restored from the location where the particle is found.

For more details on the particle tracking look at the other modules utilizing the same features in a more generic way.

35.2 Theory

In particle advection we assume that the fields are transported diffusion-free carried by a velocity field \vec{v} . The particles are initialized at the nodes of the mesh. Thereafter each particle is followed $-\delta t$ in time i.e. the following integral is evaluated

$$\vec{r} = \vec{r}_0 + \int_0^{-\delta t} \vec{v} dt. \quad (35.1)$$

Currently the integral may be evaluated using first order explicit scheme or a second order Runge-Kutta scheme. In the first order scheme a quadratic correction term is available making the scheme effectively comparable with the Runge-Kutta scheme. There the following approximation is used

$$\langle \vec{v} \rangle = \vec{v}_0 + \frac{1}{2}(\nabla \vec{v}_0) \cdot \vec{v}_0 dt. \quad (35.2)$$

When the particles have been transported the field may be evaluated from

$$f(\vec{r}_0, t) = f(\vec{r}, t - \delta t) \quad (35.3)$$

The treatment of boundaries results to some additional complication. It is assumed that if the particles vanishes in the upstream boundary then the boundary values of f are used.

The particles may have some properties along the path integral. For this purpose the user may evaluate evolution over time,

$$I_t = \int c(t) dt \quad (35.4)$$

and over distance,

$$I_s = \int c(t) ds \quad (35.5)$$

35.3 Parallel operation

The parallel operation of the particle advector routine is much more complicated than the serial. The particles are followed in the partitioned mesh and if they pass the partition interface they are passed to the next partition. Finally the values must be sent back to the originating partition in order to collect the results.

The 2nd order Runge-Kutta method will probably have problems in parallel so the user should rather choose the quadratic correction method which offers similar accuracy.

35.4 Keywords

Solver `solver id`

Equation `String [ParticleAdvectord]`

The name of the equation.

Procedure `File "ParticleAdvectord" "ParticleAdvectord"`

The name of the procedure.

Coordinate Initialization Method `String`

This is automatically enforced to nodal ordered.

Particle Node Fraction `Real`

This is automatically enforced to one in order to have one particle for each node.

Initialization Mask Variable `String`

If this is given then the particles are initialized only in nodes where the variable has a positive value.

Velocity Initialization Method `String`

This is enforced to nodal velocity which means that the first velocity is taken from the nodal point.

Velocity Variable Name `String`

Name of the velocity field in vector form. Default is flow solution.

Time Order `Integer`

This is defaulted to zero which means that the velocity field is used directly for the particle velocity.

Timestep Size `Real`

There are several keywords related to the timestepping strategy. All of them are available also here. The negative sign is added internally. Here just the most typical ones are given. This keyword gives the internal timestep size.

Simulation Timestep Sizes `Logical`

Alternatively, one may use the timesteps as defined by the Timestep Sizes of the

Max Timestep Intervals `Integer`

Maximum number of internal timesteps.

Particle Accurate At Face `Logical`

When hitting the well this keyword enforces the more accurate integration method which returns the correct point of exit. This is then used in the advection as the point of evaluation. When using accurate particle detection the algorithms might not always be as robust.

Runge Kutta Logical

Use the 2nd order Runge-Kutta method for integration.

Velocity Gradient Correction Logical

This is an alternative way of increasing the accuracy of the integral. Here the gradient of the velocity field is evaluated at the point of the particle to account for the curvature of the flow.

Source Gradient Correction Logical

This is a way to increase accuracy of the path integrals by evaluating the integrands $c(t)$ and $c(s)$ using gradient correction over the step.

Variable i String

Names of the variables to be advected ($i=1,2,3,\dots$). Any proper field variable of Elmer may be advected. The field may exist in advance, if not it will be created. There is also a group of internal variables with fixed name: particle status, particle number, particle distance, particle coordinate, particle coordinate_abs, particle velocity, particle velocity_abs, particle time, particle time integral, and particle distance integral. These are related to the particle tracking machinery.

Result Variable i String

The default name of the advected variable is obtained by adding the prefix Adv to the field name. Alternatively, the user may give the name of the result variable by this keyword.

Operator i String

Possible operator that may be applied to the variable. The choices are derivative (with respect to time), difference, and cumulative. By default no additional operator is applied.

Norm Variable Index Integer

The solver may compute the change in one specific field value in order to provide information for consistency check, or convergence monitoring. This keyword sets the index related to the Variable i list. Default is zero i.e. no norm is computed.

Particle Info Logical

Show the particle information at the end of the solver execution.

Body Force bf id

Particle Distance Integral Source T

The integrand $c(s)$ related to the distance path integral. Existence of this keyword will activate also the path integral variable named Particle Distance Integral.

Particle Time Integral Source T

The integrand $c(t)$ related to the time path integral. Existence of this keyword will activate also the path integral variable named Particle Time Integral.

Boundary Condition bc id

Particle Wall T

This will mark the wall which the particle cannot go through and by default stops at.

Part VII

Mesh Adaptation and Transformation

Model 36

Mesh Adaptation Solver

Module name: MeshSolve

Module subroutines: MeshSolver

Module authors: Juha Ruokolainen

Document authors: Juha Ruokolainen

Document edited: April 5th 2002

36.1 Introduction

Moving boundaries are often encountered in different types of computations, i.e. Fluid-Structure-Interaction (FSI) problems. Moving boundaries pose the problem of mesh adaptation to the boundaries. With this solver, instead of generating the whole mesh afresh when a boundary is moved, the current mesh nodes are moved so that the mesh hopefully remains 'good'. This type of solution only applies to cases where the changes in geometry are relatively small. It is, however, often cheaper in terms of CPU time to use this module in contrast to regenerate the whole mesh.

For time dependent simulations the mesh deformation velocity is also computed. The name of this variable is `Mesh Velocity`.

36.2 Theory

The equation for elastic deformation of the mesh, given displacement of the boundaries, may be written as

$$-\nabla \cdot \tau = 0, \quad (36.1)$$

where, \vec{d} is the mesh displacement field and τ the stress tensor.

The stress tensor given in terms of Lamé parameters is:

$$\tau = 2\mu\varepsilon + \lambda\nabla \cdot \vec{d}I \quad (36.2)$$

where μ and λ are the first and second Lamé parameters respectively, and I is the unit tensor. The linearized strains are given as:

$$\varepsilon = \frac{1}{2}(\nabla \vec{d} + (\nabla \vec{d})^T). \quad (36.3)$$

Lamé parameters in terms of Young's modulus and Poisson ratio read

$$\mu = \frac{Y\kappa}{(1-\kappa)(1-2\kappa)}, \quad \lambda = \frac{Y}{2(1+\kappa)} \quad (36.4)$$

Quantities Y and κ are the Young's modulus and Poisson ratio respectively. Note that in this context the values of the material parameters are fictional, and may be chosen to help convergence or quality of the resulting mesh.

36.2.1 Boundary Conditions

For each boundary a Dirichlet boundary condition

$$d_i = d_i^b \quad (36.5)$$

may be given. Usually this the displacement is given a priori or computed by, for example, the elasticity solvers.

36.3 Keywords

`Solver solver id`

Note that all the keywords related to linear solver (starting with `Linear System`) may be used in this solver as well. They are defined elsewhere.

`Equation String [Mesh Update]`

The name of the equation. If different from the default name `Mesh Update` then the following two keywords must be defined as well.

`Procedure File "NonphysicalMeshSolve" "NonphysicalMeshSolver"`

Name of the solver subroutine.

`Variable String`

Name of the variable.

`Equation eq id`

The equation section is used to define a set of equations for a body or set of bodies:

`Mesh Update Logical`

if set to `True`, solve the mesh adaptation equations.

`Material mat id`

The material section is used to give the material parameter values. The following material parameters may be set in Navier equations.

`Poisson Ratio Real`

For isotropic materials Poisson ratio must be given with this keyword.

`Youngs Modulus Real`

The elastic modulus must be given with this keyword.

`Boundary Condition bc id`

The boundary condition section holds the parameter values for various boundary condition types. Dirichlet boundary conditions may be set for all the primary field variables. The one related to Navier equations are

`Mesh Update i Real`

Dirichlet boundary condition for each displacement component $i = 1, 2, 3$. The boundary displacement may be computed some other solver. The computed displacement field then may be used in the setting in the following way:

`Mesh Update i Equals Displacement i` with $i = 1, 2, 3$. Including such lines in the boundary condition setting will give the mesh update on the boundary directly from the displacement solver.

36.4 Examples

36.4.1 A Simple FSI computation using MeshSolver

In this simple computation Navier-Stokes equations are solved in the domain shown in the two pictures below. On the left there is an inflow boundary, and on the right an outflow boundary. In the block inside the flow domain (the mesh is not shown for the block), the elasticity equations are solved. The block is fixed at the bottom, and is otherwise deformed by the fluid pressure and flow fields. The whole system is iterated as follows:

- Solve fluid flow,
- Solve deformation of the block,
- Solve the fluid domain mesh with MeshSolver according to the displacements of the block,

until convergence is obtained.

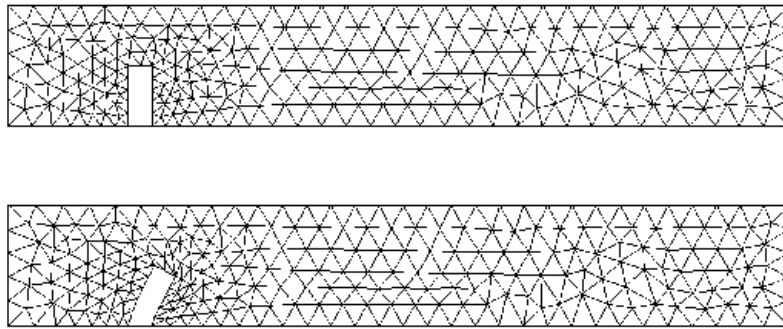


Figure 36.1: The original computational mesh (up), and the mesh of the converged solution (down) of a FSI computation.

Model 37

Nonphysical Mesh Adaptation Solver

Module name: NonphysicalMeshSolve

Module subroutines: NonphysicalMeshSolver

Module authors: Juha Ruokolainen, Peter Råback

Document authors: Juha Ruokolainen, Peter Råback

Document edited: 6th Sep 2010

37.1 Introduction

This solver is a variation of the `MeshSolver` for cases where the true mesh velocity is not of concern and more liberties can be used in the mesh adaptation. Also it may be used as the mesh adaptation solver in conjunction with `MeshSolver`. For example, in shape optimization of fluid-structure interaction problems two mesh adaptation solvers may be needed simultaneously.

37.2 Theory

For the equation to be solved look at the theory section of the `MeshSolver`. In addition to that, also weak ways of giving boundary conditions is implemented, namely

$$\tau \cdot \vec{n} = kd + f + c(d - d_0) \quad (37.1)$$

where k is a spring coefficient, f is a given force, and d_0 is the target configuration. When c goes to infinity this condition approaches the Dirichlet conditions.

37.3 Keywords

`Solver` `solver id`

Note that all the keywords related to linear solver (starting with `Linear System`) may be used in this solver as well. They are defined elsewhere.

`Procedure` `File` "NonphysicalMeshSolve" "NonphysicalMeshSolver"

Name of the solver subroutine.

`Variable` `String` [-dofs 3 Mesh Deform]

The name of the displacement field. It should be different from `Mesh Update` in order to avoid conflicts in its interpretation. Here we use the name `Mesh Deform`. The dimension should be the same as that of the mesh.

`Cumulative Displacements` `Logical`

If the same solver is called multiple times then this flag controls whether the displacements are added each time to the initial or previous mesh shape. The default is `False`.

Moving Mesh *Logical*

This keyword relates to a mesh that is being moved by an outside solver such as the `MeshSolver`.
The default is `True`.

Target Field *String*

The name of the field d_0 that is used as a target when setting the boundary conditions in a weak manner.

Nodal Penalty Factor *Real*

A coefficient that is used to set the displacements to that given by the target field in a soft manner.
This is constant for each node which results to problems in mesh consistency.

Material *mat id*

The material section is used to give the material parameter values. The following material parameters may be set in Navier equations.

Poisson Ratio *Real*

For isotropic materials Poisson ratio must be given with this keyword.

Youngs Modulus *Real*

The elastic modulus must be given with this keyword.

Boundary Condition *bc id*

The boundary condition section holds the parameter values for various boundary condition types. Dirichlet boundary conditions may be set for all the primary field variables. The one related to Navier equations are

Mesh Deform *i Real*

Dirichlet boundary condition for each displacement component $i = 1, 2, 3$.

Mesh Coefficient *i Real*

The spring coefficient related to the given coordinate direction, $i = 1, 2, 3$.

Mesh Force *i Real*

The right-hand-side of the mesh deformation equation, $i = 1, 2, 3$.

Mesh Normal Force *Real*

The right-hand-side of the mesh deformation equation in the normal direction.

Mesh Penalty Factor *Real*

When using the soft way of setting boundary conditions this value gives the weight function c .

Model 38

Rigid Mesh Transformation

Module name: RigidMeshMapper
Module subroutines: RigidMeshMapper
Module authors: Peter Råback
Document authors: Peter Råback
Document created: 10.9.2009
Document edited: 23.5.2019

38.1 Introduction

Sometimes there is a need to transform meshes without generating a new mesh. The most simple case is that of a rigid motion where some bodies move according to prescribed rotations and translations. Typically this could be a preprocessing step in a parametric study of some problem. Then this solver may be used to perform the mesh transformation.

In addition to applying rigid transformations to bodies followed by a stretching, this solver includes also a relaxation parameter which may be used to define which fraction of the node coordinates is taken from the suggested coordinates, and which part from the original coordinates.

It should be noted that the usage of this solver is rather limited. It cannot handle cases where bodies move with respect to one another if there is a mesh between the bodies. Then the `MeshUpdate` solver should be used instead.

38.2 Theory

Given original coordinate \vec{x}_0 the solver applied first a rotation, then a translation, and finally a scaling operator such that the suggested new coordinates yield

$$\vec{x}_1 = \mathcal{S}(\mathcal{R}(\vec{x}_0 - \vec{o}) + \vec{t}) + \vec{o}, \quad (38.1)$$

where \vec{t} is the vector of translation, \vec{o} is the origin, \mathcal{S} the scaling matrix, and \mathcal{R} is the rotation matrix. Rotation may currently be performed only around one main axis.

Often it is desirable that the rigid transformations are performed only for some objects and while some stay fixed. Between them the transformation degree should vary smoothly. To this aim the solver may be used to compute a degree of transformation field from the Poisson equation

$$-\nabla \cdot (1 + c|\nabla\Phi|)\nabla\Phi = s \quad (38.2)$$

where c is an optional coefficient which may be used to increase the mesh rigidity around singularities. A suitable boundary condition is $\Phi = 0$ for fixed objects and $\Phi = 1$ for moving objects.

Usually the rigid mesh mapper does not have a source term. However, for different testing purposes there is also the possibility to give a source term s which may be used to distort the mesh in a continuous way. Upon request the deformation may then be normalized to unity.

When the rigid mesh mapping is applied together with the relaxation, the end result is

$$d\vec{x} = \Phi(\vec{x}_1 - \vec{x}_0). \quad (38.3)$$

38.3 Keywords

Solver `solver id`

Equation `String [RigidMeshMapper]`

The name of the equation.

Procedure `File "RigidMeshMapper" "RigidMeshMapper"`

The name of the procedure.

Variable `String`

Optionally the solver may be used to compute a relaxation field whose values are on the interval $[0, 1]$. The final displacement is then obtained as a product of the field and the suggested rigid body motion. The name is arbitrary since it is not referenced elsewhere.

Use Original Coordinates `Logical`

This keyword applies only to cases where the solver is called repeatedly. With this keyword being true the mesh transformation is always applied to the original coordinates. Otherwise the mesh transformation is performed recursively.

Mesh Rotation Axis Order(dim) `Integer`

The user may specify the order in which the mesh is rotated around the axes with this keyword. By default first rotation is around x -axis, then around y , and finally around z . Different order will give a different end result.

Translate Before Rotate `Logical`

If this keyword is given value True, then the translation is carried out before rotations. The default is vice versa.

Cumulative Displacements `Logical`

The displacement resulting from this solver may be either absolute or cumulative. For example, for rotating problems if the mode is cumulative only the incremental angle is given. If the cumulative mode is not enforced, then the full angle from the start of the simulation should be given. The default is False.

Calculate Mesh Velocity `Logical`

If this keyword is enforced, then the solver will compute the mesh velocity resulting from rigid body deformation in a transient case. The name of the resulting vector field will be Mesh Velocity.

Mesh Relax Normalize `Found`

Normalize the mesh relaxation field such that the maximum value is one.

Body Force `bf id`

The mesh transformations are defined in this section.

Mesh Translate `Real [txtytz]`

The translational vector which may also be given individually for each component, $i = 1, 2, 3$.

Mesh Rotate `Real [αxαyαz]`

The rotation around main coordinate directions. This may also be given individually for each component, $i = 1, 2, 3$. When given for each component, they may also be variables of time, for example.

Mesh Scale Real [$s_x s_y s_z$]

The scaling around of main directions. This may also be given individually for each component, $i = 1, 2, 3$.

Mesh Origin Real [$o_x o_y o_z$]

The origin used in rotation and scaling.

Mesh Matrix(dim, dim) Real

Give the mesh transformation matrix. This overrides all other rigid mesh mapping keywords.

Mesh Displace i Real

An alternative for giving the mesh deformation in rigid body motion. Give separately for each component, $i = 1, 2, 3$. This is a local field that may vary between the nodes while the rigid body motion may only depend on global variables such as time.

Mesh Relax Real

The relaxation factor determining which amount of the coordinate transformation is taken into account. This is a local field which may depend on coordinate values whereas the other above keywords must be constant for each body force.

Mesh Relax Source Real

An optional source term for the mesh relaxation field.

Boundary Condition bc id

The boundary conditions that define the moving and fixed walls.

Moving Boundary Logical

Gets relaxation field multiplied by one.

Fixed Boundary Logical

Gets relaxation field multiplied by zero.

Model 39

Structured Mesh Mapper

Module name: StructureMeshMapper
Module subroutines: StructureMeshMapper
Module authors: Peter Råback and Thomas Zwinger
Module status: Beta
Document authors: Peter Råback and Thomas Zwinger
Document created: 4.4.2011
Document edited: 29.7.2014

39.1 Introduction

For structured meshes some operations may be done much more effectively than for generic meshes. One such operation is mapping of the mesh between given top and bottom surfaces. For example, the mesh for the computational glaciology could be deduced from a uniform initial mesh when its top and bottom surface would be known in some given fields.

39.2 Theory

The algorithm used for the mapping has two sweeps. Assume that we would like to perform mapping in direction \vec{e}_z . At the first sweep over all elements we would deduce pairwise information over nodes on which nodes are in up and down directions from each other. Then using this directional information recursively one can easily deduce which nodes are the top and bottom representatives of any node in the mesh. With this information any node between the top and bottom surface may be mapped as the linear combination of the top and bottom displacements. In the end top is mapped to the given top position and bottom to the given bottom, correspondingly.

39.3 Keywords

Solver `solver id`
Equation `String [StructuredMeshMapper]`
The name of the equation.
Procedure `File "StructuredMeshMapper" "StructuredMeshMapper"`
The name of the procedure.
Active Coordinate `Integer`
The direction in which the stuctured mapping is performed i.e. 1, 2 or 3.

Displacement Mode Logical

The values may be either used either directly as absolute coordinate values, or as displacement adding them to the original coordinate values. With this keyword the latter may be chosen.

Dot Product Tolerance Real

When determining the structure of the mesh in the active direction this tolerance is used to decide that an element edge is aligned with the direction of the action.

Top Surface Level Real

If the value is constant, then this keyword may be used to give the top surface position.

Top Surface Variable Name String

The top surface may be given by some auxiliary variable computed by some other solver, for example.

Bottom Surface Level Real

If the value is constant, then this keyword may be used to give the bottom surface position.

Bottom Surface Variable Name String

The bottom surface may be given by some auxiliary variable computed by some other solver, for example.

Mid Surface Real

Sometimes there is a middle layer that needs to be mapped as well. Then this keyword may be used. The mapping is then done linearly in two parts.

Correct Surface Logical

If this keyword is set to True, a minimum height (see next keyword) is applied to the extrusion.

Minimum Height Real

Sets the constant minimum extrusion height.

Correct Surface Mask String

This optionally defines the existing (e.g., exported) variable name where the information on whether a point is part of a column that has been corrected to the minimum height. It is -1 if it has been corrected and +1 else.

Mesh Velocity Variable String

This keyword is used to give the variable in which mesh velocity will be computed to. The mesh velocity will be really 1D only so this variable should be a scalar. If the user wants to compute a vector then the correct component of that should be given as the parameter.

Mesh Update Variable String

This keyword is used to give the variable in which mesh coordinate will be computed to. The new coordinate will be really 1D only so this variable should be a scalar. If the user wants to compute a vector then the correct component of that should be given as the parameter.

Displacement Mode Logical

The coordinates resulting from this solver may be either absolute or cumulative. If displacement mode is not enforced then the coordinates will be treated as absolute values. The default is False.

Mesh Velocity First Zero Logical

If this keyword is set True then the 1st time this routine is visited the mesh velocity is enforced to zero. May be attractive if the initial geometry is not really the initial state.

Boundary Condition bc id

Top Surface Real

The top surface position may also be given with a boundary condition.

Bottom Surface Real

The bottom surface position may also be given with a boundary condition.

Mid Surface Real

Optionally one may give a mid surface that should be between the top and bottom surfaces. This is the only method for giving the mid surface since it also specifies the active nodes whereas for top and bottom they are defined even without the boundary condition.

Part VIII

Utility Solvers

Model 40

Streamline Computation

Module name: StreamSolver

Module subroutines: StreamSolver

Module authors: Mika Juntunen, Peter Råback

Document authors: Mika Juntunen, Peter Råback

Document created: 30.7.2003

Document edited: 1.7.2011

40.1 Introduction

Streamline is a line in flow whose tangent is parallel to velocity field \vec{u} of the flow in every point \vec{x} . It should be noted that the path of material is generally not the same as streamlines. There is also third set of closely related lines, namely streak lines. On certain streak line lie all those flow elements that at some earlier instant passed through certain point in domain. Of course, the streak lines are generally different than streamlines but when the flow is steady all three set of lines coincide.

Streamlines are mainly used in providing a picture of the flow field. Drawing streamlines so that neighbouring streamlines differ by the same amount, gives a picture where direction and magnitude change of flow are clearly prescribed.

40.2 Theory

We are restricted here to the incompressible, steady flow in 2D geometry. The geometry may be 3D, but it must effectively be 2D as in axis symmetric geometry.

In 2D cartesian geometry stream function ψ is defined

$$u = \frac{\partial \psi}{\partial y}, \quad v = -\frac{\partial \psi}{\partial x}. \quad (40.1)$$

Here the geometry is (x, y) and the corresponding flow is $\vec{u} = (u, v)$. Let Ω be the domain of the flow and \vec{v} a test function for the flow. Definition (40.1) leads to finite element approximation

$$\int_{\Omega} \nabla \psi \cdot \vec{v} \, d\Omega = \int_{\Omega} \vec{u}^{\perp} \cdot \vec{v} \, d\Omega \quad (40.2)$$

In axis symmetric geometry the mass conservation calculated in a different way. This leads to following definition for stream function.

$$u = \frac{1}{r} \frac{\partial \psi}{\partial r}, \quad v = -\frac{1}{r} \frac{\partial \psi}{\partial z} \quad (40.3)$$

where the cylindrical coordinates are (z, r, ϕ) , velocity components are (u, v, w) and axis of symmetry is z i.e. $r = 0$. This function is sometimes called the *Stokes stream function* and it is not as informative as the

stream function in cartesian case. Of course the finite element approximation is a bit different.

$$\int_{\Omega} \nabla \psi \cdot \vec{v} \, d\Omega = \int_{\Omega} \vec{u}^{\perp} \cdot \vec{v} \, d\Omega \quad (40.4)$$

Here the ϕ component of the flow is excluded.

From definitions (40.1) and (40.3) it is apparent that stream function is constant along the streamlines. So drawing the contours of stream function gives the streamlines.

Sometimes setting the Dirichlet node does results to local distortion of the streamline function. To circumvent this there is an alternative way to fix the level of the streamline. There a implicit penalty $c\psi$ is added to the equation which effectively defines a finite level for the streamfunction. Small values of c are to be favored in order not to distort the streamline field. Using this penalty method the magnitude of the streamline may be quite large but after scaling the values should be rather independent on the value of c .

40.3 Limitations

Some limitations of the current implementation:

- The flow field is assumed to be incompressible.
- There is no dependency on time. Solver can be used in transient cases, but it only produces the streamlines of the current flow field as if it was steady.
- Only 2D cartesian and axis symmetric coordinate systems are implemented.
- Solver gets the velocity field from user defined variable. In cartesian case it assumes that first degree of freedom is the x -component and the second is the y -component of the velocity. In axis symmetric case it assumes that the first degree of freedom is the r -component and the second is the z -component of the velocity field.
- User can define the node whose value is first set to zero. This *shouldn't* have affect on results if the normal stream function is used in cartesian coordinates and Stokes stream function in axis symmetric coordinates. However, if used stream function is forced to something else, the position of the first node usually has a large effect on results. This is because the mass conservation is calculated differently.

40.4 Keywords

Simulation

Coordinate System String

The coordinate system should be set to be one of the following options: Cartesian 2D or Axisymmetric.

Solver solver-id

All the keywords beginning Linear System can be used. They are explained elsewhere.

Equation String

The name you want to give to the solver, for example StreamSolver.

Procedure File "StreamSolver" "StreamSolver"

The name of the file and subroutine.

Variable String

The name you want to call the solution, for example StreamFunction.

Variable DOFs Integer 1

The degree of freedom of the variable. Stream function is scalar so this must be set to 1.

Stream Function Velocity Variable String

The name of the velocity field variable. FlowSolvers solution is called Flow Solution and this is also the default value.

Stream Function First Node Integer

Number of the node that is set to zero. If given, non-positive values are set to 1 and too large values are set to largest possible i.e. 'the last node'. If not given, then other means are assumed to be used for setting the level.

Stream Function Penalty Real

When the level of the streamline is defined by a penalty formulation then this keyword is used to define the penalty factor c . Default is zero.

Stream Function Shifting Logical

Shift the smallest value to zero. Default is True.

Stream Function Scaling Logical

Scale largest absolut value to 1. Default is False.

Stokes Stream Function Logical

This keyword forces the stream function type regardless of the coordinate system. If the coordinate system is axis symmetric, then the default is True, else the default is False.

40.5 Example

This example computes the streamlines from a 2D incompressible flow field assuming the default name for flow field

Solver 4

```
Exec Solver = after all
Equation = "streamlines"
Procedure = "StreamSolver" "StreamSolver"
Variable = String Stream

Linear System Solver = "Iterative"
Linear System Iterative Method = "cg"
Linear System Preconditioning = ILU0
Linear System Residual Output = 10
Linear System Max Iterations = Integer 500
Linear System Convergence Tolerance = 1.0e-10
Linear System Abort Not Converged = False
```

End

Model 41

Flux Computation

Module name: FluxSolver

Module subroutines: FluxSolver

Module authors: Juha Ruokolainen, Peter Råback

Document authors: Peter Råback

Document edited: 21.6.2007

41.1 Introduction

This module is used to calculate the fluxes resulting usually from poisson kind of equations. These include, for example, the heat equation, the electrostatic equation, and the pressure equation for Darcy's flow. There are also flux computation subroutines that are built in the solvers but this provides a generic approach that should be easy to combine with most solvers.

41.2 Theory

Given a potential ϕ it is often interesting to know its gradient or the resulting flux. The gradient may be computed from $\nabla\phi$. The flux resulting from a potential field is assumed to be proportional to the gradient. The proportionality coefficient c may be conductivity, permeability, diffusivity etc. depending on the application field. It may be a scalar or a tensor of second kind. The flux may now be expressed as

$$\vec{q} = -c\nabla\phi. \quad (41.1)$$

For heat equation the potential would this be temperature and the conductivity would be the heat conductivity. The magnitude of a flux (or gradient) may be defined as

$$|q| = |\vec{q} \cdot \vec{q}| = |c\nabla\phi \cdot c\nabla\phi| \quad (41.2)$$

The computation of magnitude may be done before or after the numerical discretization giving result to slightly different results.

41.3 Implementation issues

The flux may be computed in many ways. Often for visualization purposes it suffices to take some nodal average of the element-wise computed fluxes. The most consistent method for flux computation is, however, using the finite element method to solve the equation (41.1). The Galerkin method creates a diagonally dominated matrix equation that may be computed easily with iterative methods even with poor preconditioners.

The flux computation may be done component-wise so that for each component q_i , where $i = 1 \dots \text{dim}$, is solved separately. This saves a significant amount of memory even though it slightly complicates the

implementation. In the solver it is also possible to choose just one component as could be sometimes desirable.

The main limitation of the current version is that it does not take into account any boundary conditions. Therefore if there is a internal boundary over which the flux is not continuous the calculated value does not make sense.

41.4 Keywords

Solver `solver id`

Equation `String Flux Solver`

Procedure `File "FluxSolver" "FluxSolver"`

Discontinuous Galerkin `Logical`

For discontinuous fields the standard Galerkin approximation enforces continuity which may be unphysical. As a remedy for this the user can enforce Discontinuous Galerkin (DG) formulation. Then the result may be discontinuous, and may even be visualized as such if the postprocessing format supports it.

Average Within Materials `Logical`

This keyword enforces continuity within the same material in the DG discretization using the penalty terms of the DG formulation.

Calculate Flux `Logical`

This flag controls the computation of fluxes. By default is `False` but its turned `True` if no other flag is not.

Calculate Flux Abs `Logical`

In conjunction with flux computation this flag may be used to compute the absolute value of the flux vector. It requires that the previous flag is active.

Calculate Flux Magnitude `Logical`

This flag computed the magnitude of the vector field. Basically it is the same in continuous level as the previous but this requires less memory and solver the matrix equation only once. The downside is that even negative values may be introduced.

Calculate Grad `Logical`

This flag turns on gradient computation. The default is `False`.

Calculate Grad Abs `Logical`

In conjunction with gradient computation this flag may be used to compute the absolute value of the flux vector. It requires that the previous flag is active.

Calculate Grad Magnitude `Logical`

This flag computed the magnitude of the vector field. Basically it is the same in continuous level as the previous but this requires less memory and solver the matrix equation only once. The downside is that even negative values may be introduced.

Enforce Positive Magnitude `Logical`

If this is active then the negative values of the computed magnitude fields are a posteriori set to zero.

Target Variable `String "Temperature"`

This gives the name of the potential variable used to compute the gradient. By default the variable is `Temperature`.

Flux Coefficient `String "Heat Conductivity"`

This gives the name of the potential variable used to compute the gradient. By default the coefficient is `Heat Conductivity`.

The solver is easily solved even without preconditioning. For example, the following linear system control may be applied.

```
Linear System Solver    "Iterative"  
Linear System Iterative Method  "BiCGStab"  
Linear System Preconditioning  None  
Linear System Max Iterations  500  
Linear System Convergence Tolerance  1.0e-10
```

41.5 Example

This example computes the diffusive heat flux in the whole domain using after the whole solution has been performed.

```
Solver 3  
  Exec Solver = after all  
  Equation = "flux compute"  
  Procedure = "FluxSolver" "FluxSolver"  
  Calculate Flux = Logical True  
  Flux Variable = String Temperature  
  Flux Coefficient = String "Heat Conductivity"  
  
  Linear System Solver = "Iterative"  
  Linear System Iterative Method = "cg"  
  Linear System Preconditioning = ILU0  
  Linear System Residual Output = 10  
  Linear System Max Iterations = Integer 500  
  Linear System Convergence Tolerance = 1.0e-10  
End
```

Model 42

Vorticity Computation

Module name: VorticitySolver
Module subroutines: VorticitySolver
Module authors: Peter Råback
Document authors: Peter Råback
Document edited: 14.2.2008

42.1 Introduction

This module is used to calculate the vorticity of vector fields. Vorticity may be of interest mainly in the post-processing of flow fields or electromagnetic fields. The default name for the vorticity is `Curl varname`.

42.2 Theory

The vorticity \vec{w} of a vector field \vec{v} is obtained simply from the curl of the field,

$$\vec{w} = \nabla \times \vec{v}. \quad (42.1)$$

Component-wise the equations for the vorticity read

$$w_x = \frac{\partial v_z}{\partial y} - \frac{\partial v_y}{\partial z} \quad (42.2)$$

$$w_y = \frac{\partial v_x}{\partial z} - \frac{\partial v_z}{\partial x} \quad (42.3)$$

$$w_z = \frac{\partial v_y}{\partial x} - \frac{\partial v_x}{\partial y}. \quad (42.4)$$

Thus, all three components exist only in 3D while in 2D and axisymmetric cases only the z -component is present

The most consistent method for computing the vorticity in conjunction with the finite elements is to solve the equations (42.4) using the Galerkin method. The resulting matrix is diagonally dominated and may be computed easily with iterative methods even with poor preconditioners. In 3D the vorticity computation may be done component-wise so that each component w_i , where $i = 1, 2, 3$, is solved separately. This saves some memory and may also save in the overall time consumption. If only one component is desired in the 3D computations then the z -component is computed.

42.3 Keywords

Solver `solver id`

```
Equation String Vorticity Solver
```

```
Procedure File "VorticitySolver" "VorticitySolver"
```

```
Target Variable String "Velocity"
```

This gives the name of the vector variable used to compute the vorticity. By default the variable is Velocity.

```
Constant Bulk Matrix Logical
```

This keyword may be used to activate the saving of the stiffness matrix if the same solver is called repeatedly. The stiffness matrix depends only on geometric information and is hence the same if the geometry is unaltered.

The solver is easily solved even without preconditioning. For example, the following linear system control may be applied.

```
Linear System Solver "Iterative"
```

```
Linear System Iterative Method "cg"
```

```
Linear System Preconditioning None
```

```
Linear System Max Iterations 500
```

```
Linear System Convergence Tolerance 1.0e-10
```

42.4 Example

This example computes the vorticity field after each timestep. Some resources are saved by reusing the same bulk matrix.

```
Solver 5
```

```
Exec Solver = after timestep
```

```
Equation = "vorticity"
```

```
Procedure = "VorticitySolver" "VorticitySolver"
```

```
Constant Bulk Matrix = True
```

```
Linear System Solver = "Iterative"
```

```
Linear System Iterative Method = "cg"
```

```
Linear System Preconditioning = ILU0
```

```
Linear System Residual Output = 10
```

```
Linear System Max Iterations = Integer 500
```

```
Linear System Convergence Tolerance = 1.0e-10
```

```
Linear System Abort Not Converged = False
```

```
End
```

Model 43

Divergence Computation

Module name: DivergenceSolver
Module subroutines: DivergenceSolver
Module authors: Peter Råback
Document authors: Peter Råback
Document edited: 20.4.2010

43.1 Introduction

This module is used to calculate the divergence of vector fields. Divergence may be of interest mainly in the postprocessing to check how well incompressibility constraints are honored.

43.2 Theory

The divergence d of a vector field \vec{v} is obtained simply from

$$d = \nabla \cdot \vec{v}. \quad (43.1)$$

The most consistent method for computing the divergence in conjunction with the finite elements is to solve the equation (43.1) using the Galerkin method. The resulting matrix is diagonally dominated and may be computed easily with iterative methods even with poor preconditioners.

43.3 Keywords

Solver `solver id`

Equation `String Divergence Solver`

Procedure `File "DivergenceSolver" "DivergenceSolver"`

Target Variable `String "Velocity"`

This gives the name of the vector variable used to compute the divergence. By default the variable is Velocity.

Constant Bulk Matrix `Logical`

This keyword may be used to activate the saving of the stiffness matrix if the same solver is called repeatedly. The stiffness matrix depends only on geometric information and is hence the same if the geometry is unaltered.

The following keywords are not usually needed as they are set by the initialization procedure of the solver.

Variable String

By default the variable is obtained from the divergence variable by adding a prefix `Div` to the field name. Naturally the name of the resulting field may also be given as desired.

The solver is easily solved even without preconditioning. For example, the following linear system control may be applied.

```
Linear System Solver "Iterative"
```

```
Linear System Iterative Method "cg"
```

```
Linear System Preconditioning None
```

```
Linear System Max Iterations 500
```

```
Linear System Convergence Tolerance 1.0e-10
```

Model 44

Scalar Potential Resulting to a Given Flux

Module name: ScalarPotentialSolver

Module subroutines: ScalarPotentialSolver

Module authors: Peter Råback

Document authors: Peter Råback

Document edited: 15.2.2008

44.1 Introduction

This module is an auxiliary solver that may be used to compute the scalar potential that results to a given flux. The flux is assumed to be a vector field resulting from some computation. This solver is the dual of the `FluxSolver`. Computing first the flux of a given potential and thereafter resolving for the potential that creates the flux should give approximately the original potential.

44.2 Theory

The flux resulting from a potential field is assumed to be proportional to the gradient of the field, ϕ . The proportionality factor is here called conductivity, c . The flux may therefore be expressed as

$$q = -c\nabla\phi. \quad (44.1)$$

For heat equation the potential would be temperature and the conductivity would be the heat conductivity.

This solver solves the equation in the reverse form, i.e. given the flux solver for the potential. In the weak formulation this is solved so that the test function is the gradient of the shape function. This results to the standard discretization of the Poisson equation.

The potential is not defined uniquely unless the level is fixed at least at one point. Therefore the user should set a Dirichlet condition at least at one node.

44.3 Keywords

`Solver` `solver id`

`Equation` `String` `ScalarPotentialSolver`

`Procedure` `File` `"ScalarPotentialSolver"` `"ScalarPotentialSolver"`

`Variable` `String` `"Scalar Potential"`

The desired name of the resulting scalar field.

Flux Variable `String`

This gives the name of the flux variable used to compute the source term. Note that this must be the name of a vector field such as `Velocity`.

Flux Coefficient `String`

This gives the name of the coefficient used in the computation of the flux. For example, in thermal analysis it would be `Heat Conductivity`. If a non-existing material parameter is given the coefficient will be assumed to be one, i.e. $c = 1$.

The equation is a Poisson type of equation and defaults for it are set to be `cg+ILU0`. If these do not suffice, other linear system options should be defined.

Boundary Condition `bc id`

Scalar Potential `Real`

The defined field variable must be set to be zero at least at one point.

Target Nodes `Integer`

The user may also define a target node on-the-fly at which the condition is set.

Model 45

System Reduction for Displacement Solvers

Module name: RigidBodyReduction

Module subroutines: RigidBody

Module authors: Antti Pursula

Document authors: Antti Pursula

Document edited: August 27th 2003

45.1 Introduction

This module is used to reduce and simplify the computation of a displacement solver when the problem includes rigid blocks. In such a case, it is often difficult for iterative solvers to find a solution for the full system, and direct solvers become obsolete when the system is large enough. The convergence and also the speed of the solution can be substantially improved when the degrees of freedom corresponding to the nodes belonging in the rigid blocks are reduced onto the 6 DOFs (3 in 2D) of the corresponding rigid body. In the module, the reduction is achieved via a projection matrix.

Additionally, the routine automatically eliminates the degrees of freedom corresponding to the Dirichlet boundary conditions. It is also possible to request the elastic regions to be extended into the rigid blocks. There is also possibility to reorder the reduced matrix elements to decrease its bandwidth.

45.2 Theory

The module starts with normally constructed matrix equation for the unknown displacements x , $Ax = b$. Let us assume that the nodes are ordered in such a way that the first n elements of the vectors correspond to the elastic parts of the structure and the remaining m elements correspond to the rigid parts of the structure. The goal is to reduce the $(n + m) \times (n + m)$ matrix A to a $(n + \alpha k) \times (n + \alpha k)$ matrix B , where k is 3 for 2D and 6 for 3D problems and α is the number of rigid blocks present. Reductions are made also for the vectors so that finally the matrix equation reads $Bu = f$.

The relation between the unknowns is

$$x = Pu, \quad (45.1)$$

where the projection matrix P ties the nodes in the rigid bodies to the same displacements in coordinate directions and the same rotations about the coordinate axis. The rotations are defined with a coordinate system whose origin is at the center of each rigid body. For the right hand sides we can write

$$f = Qb, \quad (45.2)$$

where the matrix Q sums the forces and torques present at the nodes in rigid bodies for a resultant force and torque of the center point of the corresponding rigid body. In both mappings, the rotations are linearized so the module is valid only for cases where the rotations are small.

Using these definitions, we have

$$Ax = APu = b \quad (45.3)$$

and

$$Bu = f = Qb. \quad (45.4)$$

Combining the equations gives $Bu = QAPu$ and thus

$$B = QAP. \quad (45.5)$$

With a suitable order of the rotations one can write

$$Q = P^T \equiv C, \quad (45.6)$$

and

$$B = CAC^T. \quad (45.7)$$

The matrix C has a identity matrix block of size $n \times n$ which keeps the elastic nodes intact, and a projection block of size $\alpha k \times m$.

The reduced order solution u is transformed back to the original nodes by the same mapping

$$x = C^T u. \quad (45.8)$$

45.3 Applicable cases and limitations

The module works for

- Linear steady-state problems
- Linear transient problems
- Eigen analysis
- Quadratic eigenproblems

There are following limitations:

- Rigid blocks should not have common nodes (there should be elastic nodes in between rigid blocks)
- If a Dirichlet bc is given on a node of a rigid block then the entire rigid block is assumed to be fixed in all directions

45.4 Keywords

Body `body id`

Rigid Body `Logical`
Value `True` defines the rigid body.

Solver `solver id`

The module does not need a separate solver but a call in the stress analysis, or the elasticity solver in the linear mode.

Equation `String Stress Analysis`

Variable `String Displacement`

Variable DOFs Integer

It is important to give the DOFs right, either 2 or 3 depending on the dimension.

Before Linsolve File "RigidBodyReduction" "RigidBody"

The model order reduction is performed after the matrix has been assembled but before the matrix equation has been solved. The matrix equation is modified to a smaller equation and the new equation is solved within the subroutine.

Eigen Analysis Logical

It is possible to use the model order reduction with modal analysis, as well as with static and transient cases.

Eigen System Values Integer

The number of eigen values to be computed.

Eigen System Damped Logical

Eigen System Use Identity Logical [True]

The reduction is possible also with quadratic (damped) eigenproblems.

Optimize Matrix Structure Logical

If true, the matrix structure is optimized. This feature is recommended since the reduced matrix has often very scattered structure. The optimization is performed with the Cuthill-McKee algorithm.

Reverse Ordering Logical

This flag can be used to reverse the matrix ordering if the matrix structure is optimized, resulting in reverse Cuthill-McKee ordering.

Extend Elastic Region Logical

If true, the elastic regions of the geometry are extended into the rigid block. This feature allows taking into account the bending in the joints between elastic and rigid parts.

Extend Elastic Layers Integer

Defines the number of element layers that the elastic regions are extended.

Output Node Types Logical

Writes in the ElmerPost output file a variable describing the status of each node in the geometry. The variable has value 0 for elastic nodes, -1 for rigid blocks that are fixed due to a Dirichlet boundary condition, and a positive integer for separate rigid blocks. The variable may be used to check that the reduction is performed on the right blocks, and to check how many layers the elastic regions should be extended, for example.

Additional Info Logical

If true, additional information is written about the performed tasks during the simulation.

45.5 Examples

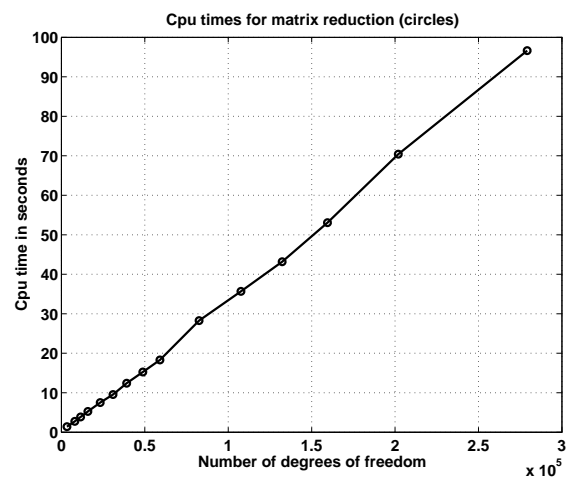


Figure 45.1: The cpu time required for the matrix reduction operations depends linearly on the degrees of freedom in the system.

Model 46

Artificial Compressibility for FSI

Module name: ArtificialCompressibility

Module subroutines: CompressibilityScale

Module authors: Peter Råback

Document authors: Peter Råback

Document created: 16.2.2002

Document edited: 8.2.2006

46.1 Introduction

When fluid-structure interaction (FSI) problems are solved with a loosely coupled iteration strategy there is a risk of applying unphysical boundary conditions that lead to severe convergence problems. The reason for this is that initially the fluid domain is unaware of the constraint of the structural domain, and vice versa. If the iteration converges this discrepancy will be settled, but sometimes the initial phase is so ill posed that convergence is practically impossible to obtain [4, 3].

The problem may be approached by applying the method of artificial compressibility to the fluid-structure interaction. Previously artificial compressibility has mainly been used as a trick to eliminate the pressure from the Navier-Stokes equations or to improve the convergence of the solution procedure [2, 6, 1]. Here the compressibility is defined so that it makes the fluid imitate the elastic response of the structure.

The method is best suited for cases where there is a direct correspondence between the pressure and the volume. Inertial forces and traction forces should be of lesser importance. The method might, for example, boost up the modeling of human arteries.

46.2 Theory

46.2.1 Fluid-structure interaction

The theoretical model with some results is thoroughly presented in

We look at the time-dependent fluid-structure interaction of elastic structures and incompressible fluid. The equations of momentum in the structural domain is

$$\rho \frac{\partial^2 \vec{u}}{\partial t^2} = \nabla \cdot \tau + \vec{f} \text{ in } \Omega_s, \quad (46.1)$$

where ρ is the density, \vec{u} is the displacement, \vec{f} the applied body force and $\tau = \tau(\vec{u})$ the stress tensor that for elastic materials may be locally linearized with \vec{u} . For the fluid domain the equation is

$$\rho \left(\frac{\partial \vec{v}}{\partial t} + \vec{v} \cdot \nabla \vec{v} \right) = \nabla \cdot \sigma + \vec{f} \text{ in } \Omega_f, \quad (46.2)$$

where \vec{v} the fluid velocity and σ the stress tensor. For Newtonian incompressible fluids the stress is

$$\sigma = 2\mu\varepsilon(\vec{v}) - pI, \quad (46.3)$$

where μ is the viscosity, $\varepsilon(\vec{v})$ the strain rate tensor and p the pressure. In addition the fluid has to follow the equation of continuity that for incompressible fluid simplifies to

$$\nabla \cdot \vec{v} = 0 \text{ in } \Omega_f. \quad (46.4)$$

For later use we, however, recall the general form of the continuity equation,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \text{ in } \Omega_f. \quad (46.5)$$

The fluid-structure interface, Γ_{fs} , must meet two different boundary conditions. At the interface the fluid and structure velocity should be the same,

$$\vec{v}(\vec{r}, t) = \vec{u}(\vec{r}, t), \quad \vec{r} \in \Gamma_{fs}. \quad (46.6)$$

On the other hand, the surface force acting on the structure, \vec{g}_s , should be opposite to the force acting on the fluid, \vec{g}_f , thus

$$\vec{g}_s(\vec{r}, t) = -\vec{g}_f(\vec{r}, t), \quad \vec{r} \in \Gamma_{fs}. \quad (46.7)$$

A widely used iteration scheme in FSI is the following: First, assume a constant geometry and solve the Navier-Stokes equation for the fluid domain with fixed boundary conditions for the velocity. Then calculate the surface forces acting on the structure. Using these forces solve the structural problem. Using the resulting displacement velocities as fixed boundary conditions resolve the fluid domain. Continue the procedure until the solution has converged.

The above described iteration usually works quite well. However, in some cases the boundary conditions (46.6) and (46.7) lead to problems. The elasticity solver is not aware of the divergence free constraint of the velocity field. Therefore the suggested displacement velocities used as boundary conditions may well be such that there is no solution for the continuity equation. A proper coupling method makes the solution possible even if the velocity boundary conditions aren't exactly correct. Further, if the Navier-Stokes equation is solved without taking into account the elasticity of the walls, the forces in equation (46.7) will be exaggerated. The pathological case is one where all the boundaries have fixed velocities. Then even an infinitely small net flux leads to infinite pressure values. A proper coupling method should therefore also give realistic pressure values even with inaccurate boundary conditions. The method of artificial compressibility meets both these requirements.

46.2.2 Artificial compressibility

When a surface load is applied to an elastic container it results to a change in the volume. In many cases of practical interest the change in volume is mainly due to a pressure variation from the equilibrium pressure that leads to zero displacements. If the structural domain is described by linear equations the change in volume dV has a direct dependence on the change in the pressure, dP , or

$$\frac{dV}{V} = c dP. \quad (46.8)$$

This assumption limits the use of the model in highly nonlinear cases.

The change in the volume should be the same as the net volume flux into the domain. As this cannot be guaranteed during the iteration, some other way to enable the material conservation must be used. A natural choice is to let the density of the fluid vary so that it has the same pressure response as the elastic walls,

$$\frac{d\rho}{\rho} = c dP, \quad (46.9)$$

where c is the artificial compressibility. This is interpreted locally and inserted to the continuity equation (46.5) while neglecting the space derivative of the density, thus

$$c \frac{dp}{dt} + \nabla \cdot \vec{v} = 0, \quad (46.10)$$

where dp is the local pressure change. Here the time derivative of pressure must be understood as an iteration trick. A more precise expression is

$$\frac{c}{\Delta t} \left(p^{(m)} - p^{(m-1)} \right) + \nabla \cdot \vec{v}^{(m)} = 0, \quad (46.11)$$

where m is the current iteration step related to fluid-structure coupling. When the iteration converges $p^{(m)} \rightarrow p^{(m-1)}$ and therefore the modified equation is consistent with the original one. The weak form of the equation for finite element method (FEM) may easily be written,

$$\int_{\Omega_f} (\nabla \cdot \vec{v}^{(m)}) \varphi_p d\Omega + \frac{1}{\Delta t} \int_{\Omega_f} c \left(p^{(m)} - p^{(m-1)} \right) \varphi_p d\Omega = 0, \quad (46.12)$$

where φ_p is the test function.

The artificial compressibility may be calculated analytically in simple geometries. For example, for a thin cylinder with thickness h and radius R the compressibility is $c = 2R/Eh$ [5], where E is the Young's modulus, and correspondingly for a sphere $c = 3R/Eh$.

In most practical cases the elastic response of the structure cannot be calculated analytically. Then the compressibility may also be computed from equation (46.8) by applying a pressure change dP to the system,

$$c = \frac{1}{V} \frac{dV}{dP}. \quad (46.13)$$

The change in volume may be calculated by comparing it to initial volume, thus

$$c = \frac{V - V_0}{V_0} \frac{1}{dP}. \quad (46.14)$$

For small deformations $ds = \vec{u} \cdot \vec{n}$, where \vec{n} is the surface normal. Therefore we may use an alternative form convenient for numerical computations,

$$c = \frac{\int_{\Gamma_{fs}} (\vec{u} \cdot \vec{n}) dA}{\int_{\Omega_f} dV} \frac{\int_{\Gamma_{fs}} dA}{\int_{\Gamma_{fs}} dp dA}. \quad (46.15)$$

This way c has a constant value over the domain.

46.2.3 Scaling artificial compressibility

If the artificial compressibility distribution is a priori defined we may use the above equations to scale the compressibility appropriately. For example, the compressibility could be given only within a limited distance from the elastic wall. and the functional behavior of $c(\vec{r})$ would be user defined. Computing compressibility becomes then just a matter of scaling,

$$c(\vec{r}) = c_0(\vec{r}) \underbrace{\frac{\int_{\Gamma_{fs}} (\vec{u} \cdot \vec{n}) dA}{\int_{\Omega_f} c_0(\vec{r}) dV} \frac{\int_{\Gamma_{fs}} dA}{\int_{\Gamma_{fs}} dp dA}}_{\text{scaling factor}}. \quad (46.16)$$

A suitable test load for computing compressibility is the current pressure load on the structure. However, for the first step the compressibility must be predefined. It is safer to over-estimate it since that leads to too small a pressure increase. Too large a pressure increase might ruin the solution of the elasticity solver and by that also the computational mesh used by the flow solver would be corrupted. Therefore some sort of exaggeration factor exceeding unity might be used to ensure convergence.

46.2.4 Elementwise artificial compressibility

If the displacement field is extended smoothly throughout the whole geometry it may be possible to define the artificial compressibility separately for each element or node. This is particularly useful for geometries where the elastic response changes significantly. The equation is now similar to (46.14),

$$c = \frac{V^e - V_0^e}{V_0^e} \frac{1}{dP}, \quad (46.17)$$

where the superscript e refers to the volume of an element. This may also be solved using finite element strategies to get nodal values for c .

46.3 Keywords

Keywords of FlowSolve

Material `mat id`

In the material section the compressibility model and the initial artificial compressibility field is given.

Compressibility Model `String [Artificial Compressible]`

Set the material model of the fluid.

Artificial Compressibility `Real`

The initial value of artificial compressibility. This may also be a distributed function that is then scaled by the solver.

Keywords of solver CompressibilityScale

If the artificial compressibility is tuned so that it best imitates the elastic response, an additional solver must be used to rescale the above mentioned compressibility. The solver computes the total compressibility and the force acting on the surface. The compressibility is integrated over all volumes that are solved with the Navier-Stokes equation.

Solver `solver id`

Equation `String CompressibilityScale`

The name of the solver.

Procedure `File "ArtificialCompressibility"`

`"CompressibilityScale"`

The subroutine in the dynamically linked file.

Steady State Convergence Tolerance `Real`

How much the relative value of the compressibility may change between iterations, $\text{abs}(c_i - c_{i-1})/c_i < \varepsilon$.

Nonlinear System Relaxation Factor `Real`

Relaxation scheme $c'_i = \lambda c_i + (1 - \lambda)c_{i-1}$ for the compressibility. By default is $\lambda = 1$.

Boundary Condition `bc id`

Force BC `Logical`

The elastic response is calculated over the surface(s) which has this definition as True.

Keywords of solver CompressibilitySolver

When the compressibility is solved elementwise using this solver there has to usually be an isobaric steady-state test phase where the compressibility is defined. For this solver all the normal Linear System keywords also apply.

Solver solver id

Equation String CompressibilitySolver

Procedure File "ArtificialCompressibility"
"CompressibilitySolver"

Variable String ac

The name of the artificial compressibility field variable.

Displacement Variable Name String "Mesh Update"

The name of the displacement field variable that is used to compute the the volume change.

Displaced Shape Logical True

Flag that defines whether the current shape is the displaced or original shape.

Reference Pressure Real

The value of pressure used for the test loading.

The computed field should then be given as the value in the material section.

Material mat id

Artificial Compressibility Equals ac

The initial value of artificial compressibility given by the solver.

46.3.1 Examples

The examples show a 2D square and a 3D cube being gradually filled. The fluid comes in from one wall and the opposing elastic wall makes room for the fluid so that the continuity equation is satisfied. Here the value of artificial compressibility is scaled every timestep to account for the nonlinear elasticity.

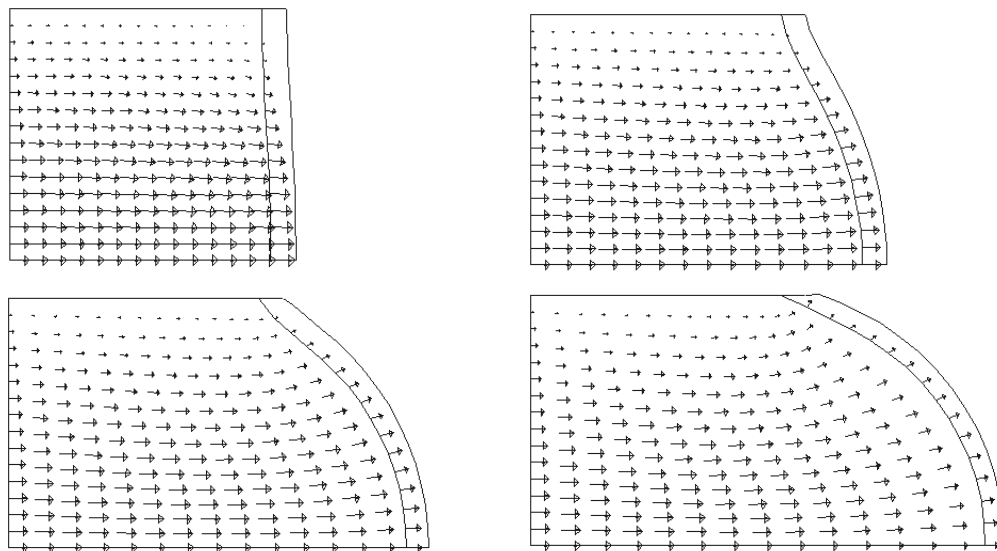


Figure 46.1: Snapshots of an elastic square being gradually filled by incompressible fluid.

Bibliography

- [1] F. D. Carter and A. J. Baker. Accuracy and stability of a finite element pseudo-compressibility cfd algorithm for incompressible thermal flows. *Num. Heat Transfer, Part B*, 20:1–23, 1991.

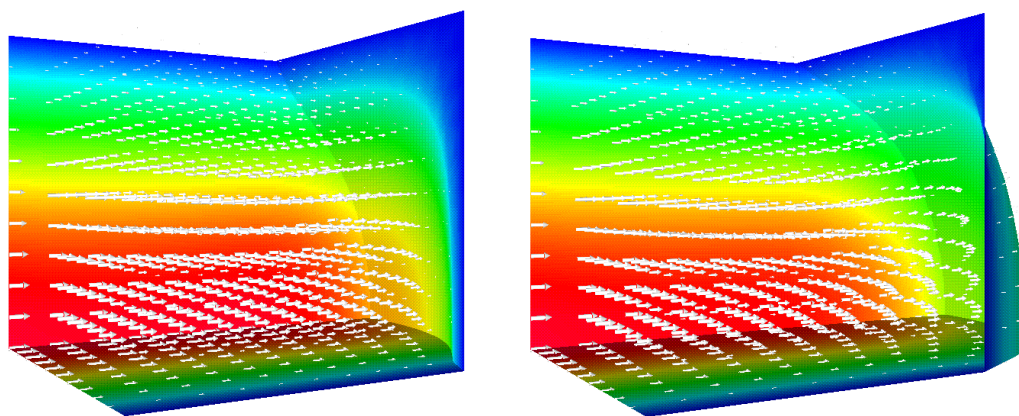


Figure 46.2: Snapshots of an elastic cube being gradually filled by incompressible fluid.

- [2] A. J. Chorin. A numerical method for solving incompressible viscous flow problems. *J. Comput. Phys.*, 135:118–125, 1997.
- [3] L. Formaggia, J. F. Gerbeau, F. Nobile, and A. Quarteroni. Numerical treatment of defective boundary conditions for the navier-stokes equations. *EPFL-DMA Analyse et Analyse Numerique*, 20, 2000.
- [4] E. Järvinen, M. Lyly, J. Ruokolainen, and P. Råback. Three dimensional fluid-structure interaction modeling of blood flow in elastic arteries. In *Eccomas Computational Fluid Dynamics Conference*, September 2001.
- [5] Kris Rienslagh, Jan Vierendeels, and Erik Dick. An efficient coupling procedure for flexible wall fluid-structure interaction. In *Eccomas Congress on Comp. Meth. in Appl. Sci. and Eng*, September 2000.
- [6] S. E. Rogers, D. Kwak, and U. Kaul. On the accuracy of the pseudocompressibility method in solving the incompressible navier-stokes equations. *Appl. Math. Modelling*, 11:35–44, 1987.

Model 47

Internal Cost Function Optimization

Module name: FindOptimum
Module subroutines: FindOptimum
Module authors: Peter Råback
Module status: Alpha
Document authors: Peter Råback
Document created: 18.3.2009
Document edited: 18.3.2009

47.1 Introduction

This solver is an auxiliary solver for optimization problems. As input it requires a cost function computed with the previous parameter values, and as output it gives the new parameters for which the cost function will be computed for. Typically the cost function depends on the solution of one or several differential equations. Based on this solution a measure of goodness for the solution is computed.

The routine is still in its development phase but is provided as a skeleton that may be further developed.

47.2 Theory

The optimization routines must be slightly modified from their standard form since the solver is not in a ruling position in respect to the simulation. Therefore it's difficult to plug in existing optimization packages to this solver.

Currently the solver includes some very basic optimization routines. Of these the Simplex algorithm (Nelder-Mead) and the differential GA (Genetic algorithm) are the only ones that may be used for a number of design variables.

For just one design variable there is the choice of simple scanning, bisection search and the secant method. Secant method finds roots making it better suited for problems where the target is known i.e. design problems.

47.3 Keywords

Simulation

Simulation Type String "scanning"

The natural mode used for optimization problems is scanning. If the problem is really time-dependent the current internal solution is not probably the optimal solution.

Timestep Intervals Integer

The maximum number of optimization rounds is in the case of scanning defined by the timestep intervals.

Solver solver id

Equation String FindOptimum

A describing name for the solver.

Variable String OptPar

The name of the variable may be freely chosen as far as it is used consistently also elsewhere.

Variable DOFs Integer n

Degrees of freedom for the pressure. Here n should be equal to the number of parameters.

Variable Global Logical True

Indicates the variable is a global one i.e. not a field variable. For global variables the number of unknowns is the same as number of dofs.

Procedure File "FindOptimum" "FindOptimum"

The name of the module and procedure. These are fixed.

Optimization Method String

Choices are currently random, scanning, genetic, bisection, secant and simplex.

Cost Function Name String

The name of the cost function that is a real stored in the Simulation list structure.

Optimal Restart Logical

Use the previous best set of parameters for the 1st round of cost function computation.

Optimal Finish Logical

Use the best set of parameters for the last round of cost function computation. This may be useful as the last step is often also saved.

Best File File [best.dat]

The file where the best set of parameters is always saved.

Guess File File [best.dat]

The file where the best set of parameters is read in case of optimal restart.

Fixed Parameter i Logical

Is the i:th parameter fixed. Applies for some optimization routines.

Min Parameter i Real

Minimum value for i:th parameter. Applies for some optimization routines.

Max Parameter i Real

Maximum value for i:th parameter. Applies for some optimization routines.

Initial Parameter i Real

Initial value for i:th parameter if not given by the Optimal restart

Internal history Logical

Save the internal values within the solver.

History File File

The name of the file where the history data is saved.

Cost Function Target Real

If the given cost function is C use $C - C_0$ instead.

Cost Function Absolute Logical

If the given cost function is C use $|C|$ instead.

Cost Function Maximize Logical

If the given cost function is C use $-C$ instead.

The following keywords apply to the GA algorithm

```
Population Size Integer [5n]
Population Coefficient Real [0.7]
Population Crossover Real [0.1]
```

The following keywords apply to the simplex algorithm

```
Simplex Relative Length Scale Real [0.01]
    The relative length scale that determines the size of the 1st simplex.
Simplex Restart Interval Integer
    The restart interval after which the simulation is restarted if the convergence is poor.
Simplex Restart Convergence Ratio Real
    A critical value which is used to define a poor convergence ratio.
```

The following keywords apply to the secant method

```
Step Size Real
    The step size of the first computations.
Max Step Size M
    Maximum allowed step size.
Relaxation Factor R
    Relaxation used in the secant method.
```

This shows just a couple of examples how the design parameters could be used in the simulation. The variables may be referred in a similar manner as other global variables such as `time` or `timestep` size.

```
Body Force 1
    Heat Source = Equals OptPar 1
End

Boundary Condition 1
    Heat Flux = Equals OptPar 2
End
```

Part IX

Saving Data and Results

Model 48

Saving Scalar Values to a File

Module name: SaveData

Module subroutines: SaveScalars

Subroutine authors: Peter Råback

Document authors: Peter Råback

Document created: Oct 3rd 2002

Document updated: January 8th 2008

48.1 Introduction

This subroutine may be used to compute derived quantities and saving scalar values to external file. The results are easily then utilized by MatLab, Excel or any other program that can read ASCII data. In addition to the number values also an additional file with the suffix `.name` is saved. It tells what variables are at each column.

48.2 Theory

The equations and algorithms needed for the computation of scalar values are relatively simple. Here some of them are introduced.

When saving statistical information there are two possibilities. We may use normal number statistics where each node is given an equal weight. Then, for example the mean becomes,

$$\langle f \rangle = \frac{\sum_{i=1}^n f_i}{n}. \quad (48.1)$$

The other possibility is to treat the variable as a continuous function and compute the statistical values as averages over the domain. Now the mean is

$$\langle f \rangle = \frac{\int f d\Omega}{\int d\Omega}. \quad (48.2)$$

In addition to the mean we may compute the mean deviation, $\langle |f - \langle f \rangle| \rangle$, and the variance $\delta f = \sqrt{\langle f^2 \rangle - \langle f \rangle^2}$.

It is possible to compute energy type of lumped quantities by integrating over the domain. The energy of the field f resulting from a diffusion equation is

$$E_{diff} = \frac{1}{2} \int_{\Omega} \nabla f \cdot c \nabla f d\Omega, \quad (48.3)$$

where c may be a tensor or a scalar. Kinetic energy related to convection is of type

$$E_{con} = \frac{1}{2} \int_{\Omega} c \vec{v} \cdot \vec{v} d\Omega, \quad (48.4)$$

and potential type of energy

$$E_{pot} = \int_{\Omega} c f d\Omega. \quad (48.5)$$

Sometimes it may be interesting to compute the fluxes through surfaces. The values may be used in evaluating the accuracy of the results – what goes in should in steady state also come out. There are two different fluxes that may be computed. For convective field the flux is of type

$$F_{con} = \int_{\Gamma} c f \vec{v} \cdot \vec{n} d\Gamma. \quad (48.6)$$

Currently the code automatically detects the dimension of the given variable. If it is scalar f then \vec{v} is assumed to be the velocity computed by Navier-Stokes equation. If the given variable is vector then it is used as the velocity field and f is not present. Here \vec{n} is the surface normal.

Diffusive fluxes may be computed from

$$F_{diff} = \int_{\Gamma} c \nabla f \cdot \vec{n} d\Gamma, \quad (48.7)$$

where c may also be a tensor.

48.3 Implementation issues

There are many solvers that internally compute lumped quantities. By convention these are added to the list structure of the `Simulation` section in the following style

```
CALL ListAddConstReal( Model % Simulation, 'res: capacitance', Capacitance)
```

By default the solver looks through quantities starting with prefix `res:` and saves them to an external file. Also the user may create new quantities to be saved in similar manner.

48.4 Keywords

`Solver solver id`

`Procedure File "SaveData" "SaveScalars"`

`Filename String`

Name of the file where results are to be saved. Of no filename is given then the results will only be echoed on output.

`Output Directory String`

Name of the directory where results are to be saved, relative to the case directory. By default the results are saved in the case directory.

`Scalars Prefix String`

Save constants starting with this prefix. The default is `res:`.

`Variable i String`

The names of the variables to be saved. There can be up to 99 variables. In addition to field variables there are some special variables. The scalar variables. e.g. `Time`, are saved as is. There are also variables `CPU Time` and `CPU Memory` that may be used to save execution details.

Target Variable *i* String

This is an optional keyword that for each entry computed by the subroutine may give an alternative name that is placed as a proper variable in the model. This variable may then be used similarly to `time` in functional expressions. Note that if the operator creates several output values the numbering of this is not the same as that of `Variables`. So check the output file for correct entries when in doubt.

Mask Name *i* String

If the operator is such that it can use masks then this keyword may be used to override the default mask name `Save Scalars`.

Save Points(*n*) Integer

Save the specified degrees of freedom in the *n* nodes specified.

Save Coordinates(*n*, DIM) Real

Save the degrees of freedom in the nodes nearest to the given *n* coordinates.

Exact Coordinates Logical

When this keyword is true the coordinates will be looked in an exact manner. Then the degrees of freedom are linear combinations of the node values of the element that the point belongs to.

Moving Mesh Logical

If this parameter is `True` the saved points will be defined every time the subroutine is visited. The default is `False`.

File Append Logical

If the results from consecutive rounds should be appended to the file this flag should be set to `True`. The default is `False`.

Filename Numbering Logical

If set to true a running index is added to the filename so that the next free filename is used to save the results.

Partition Numbering Logical

Optionally add the number of partitions to the filename. This makes the benchmarking more convenient since each case may use the same command file without conflicts.

Show Norm Index Integer

The user may choose to output one value of the results as the norm of the solver in a similar output syntax as `ComputeChange` shows its norms. This is of course not a real norm but may be used in monitoring desired convergence measures in ElmerGUI, for example. By default no norm is shown.

Echo Values Logical

When this is turned on the scalar values will also be echoed to the screen. This is the default action also when the filename is not given and hence nothing is saved to file.

Cost Function Index T

The user may also choose to save a desired value in the list structure with the name `Cost Function`. This may be utilized by the `FindOptimum` solver in optimization problems.

Parallel Reduce Logical

By default the output is written independently for each partition in parallel runs. Enabling this, however, the information is reduced to just one file. The reduction is done using `MPI_ALLREDUCE`: `MPI_MAX`, `MPI_MIN`, `MPI_SUM`. The default is `MPI_SUM`. Parallel operator is controlled with keyword `Parallel Operator`. These are not sufficient for all operators. By default the value of the 1st partition is written.

Save Eigenvalues Logical

Save the eigenvalues found in any of the variables.

Save Eigenfrequencies Logical

Save the frequencies computed from the eigenvalues found in any of the variables.

Operator i String

There are different operators that may be performed on all the given variables. These include statistical operators working on the set of numbers, `max`, `min`, `max abs`, `min abs`, `mean`, `rms`, `variance` and `deviation`. Note that these operate directly on the result vector and do not employ the mesh in anyway.

Different scalar quantities are obtained also by domain integral operators over the mesh. Operator `int` gives the integral over a variable, `int mean` the mean value, `int rms` the root mean square, and `int variance` the variance of the variable. The volume used by a given variable is obtained by operator `volume`. If a name for the coefficient, is given for the operator, the integral is taken over the coefficient. One can for example obtain the weight from a integral over `Density`. Three different energy type of energy quantities may be computed by `diffusive energy`, `convective energy`, `potential energy`, `volume`.

By default the statistical and integral operators are performed over the whole mesh. However, the user may apply the operator only to some parts of the mesh that are related to the logical entities of the `sif` file. Currently these entities are `body`, `body force`, and `material`. If the user wants to select some of these parts then the standard operator should precede by the name of the section. For example, there could be operators `body max` or `body force int`.

There are also a number of similar operators that only operate on the boundary. These are invoked by `boundary sum`, `boundary dofs`, `boundary mean`, `boundary max`, `boundary min`, `boundary max abs`, `boundary min abs`, `area`, `boundary int`, and `boundary int mean`. Also boundary integrals are possible using operators `diffusive flux`, `convective flux`, `boundary int`, `boundary int mean` and `area`. These require that in the boundary conditions the active boundaries are defined. Also here there may be an optional coefficient.

Some operators do not work on the solution itself but use other info related to that. Operator `dofs` simply returns the length of the variable under study. Operator `norm` returns the last computed norm of the field variable, and operators `nonlinear change` and `steady state change` return the last computed convergence measures at the nonlinear and steady state levels. Operator `nonlin iter` returns the number of nonlinear iterations, while operators `nonlin converged` and `coupled converged` which tell whether or not the simulation has converged. Note that these operators most operate on the primary variable for which the matrix equation is solved for.

Operator `bounding box` returns the minimum and maximum coordinate value of each coordinate i.e. six values in 3D mesh.

Finally for parallel runs the operator `partitions` may be usefull in creating parallel scaling results in automated manner.

There is no upper bound to the number of operators or variables. If the variable is a vector the statistics is performed on its length.

Coefficient i String

Even though only limited number of operators are given almost any energy or flux kind of quantity may be computed since the coefficient c may be defined by the user. The idea is that the same data that is already used as a material parameter can be simple referred to by its name. The coefficient may be, `Heat Conductivity`, `Permittivity`, `Density`, for example. Usually the coefficient is the same that was used in computing the field variable under integration. For the `diffusive energy` and `diffusive flux` the coefficient may even be a matrix. This parameter is optional and the default is one.

Parallel Operator i String

Sometimes the default parallel reduction method is not the desired one. Therefore the user may define the parallel reduction method by this keyword. The alternatives are `min`, `max` and `sum`.

Polyline Coordinates(n,DIM) Real

This keyword may be used to create line segments that are defined by points x_1 , y_1 , x_2 , and y_2 . For each line different kinds of fluxes trough the elements may be computed. This makes it possible, for example, to check the mass flux even though no boundary has a priori been defined.

```
Save component results Logical
    Save results arising from component sections.
```

```
Boundary Condition bc id
```

```
Save Scalars Logical
    The flag activates the computation of boundary-related information. The results are treated inde-
    pendently for each boundary. The keyword replaces the previously used Flux Integrate.
    Also if Mask Name is given this arbitrary string may be used instead.
```

If the user only wants a basic functionality of SaveScalars it is possible to let the system create automatically an instance simply by:

```
Simulation
```

```
Scalars File File
    Name of the file where the results are saved.

scalars: Keyword Type
    Any keyword with the suffix scalars: is passed to the solver instance without the suffix.
```

By default the solver is executed after saving. With the namespace it is possible to include all keywords (with the keyword type given) also in the simulation section. At some point the Simulation section will get crowded making it justified to create a separate solver instance.

48.5 Examples

In the following examples it is assumed that the SaveScalars solver gets the first free index.

Range of solution

The following example shows how to deduce after each converged timestep the minimum and maximum values of temperature to an external file.

```
Solver 2
    Exec Solver = After Timestep
    Equation = SaveScalars
    Procedure = "SaveData" "SaveScalars"
    Filename = "temprange.dat"

    Variable 1 = Temperature
    Operator 1 = min
    Operator 2 = max
    Operator 3 = int mean
End
```

Force resulting from flow solution

The following example shows how to compute the force resulting from the Navier-Stokes equation on the boundary 3. It is assumed that for the flow solver is set Calculate Loads = True and that the name of the solution vector is Flow Solution. Then the force acting on the boundary is obtained by summing up the nodal forces on the boundary.

```
Solver 2
    Exec Solver = After Timestep
    Equation = SaveScalars
    Procedure = "SaveData" "SaveScalars"
```

```
Filename = "forces.dat"

Variable 1 = Flow Solution Loads 1
Operator 1 = boundary sum
Variable 2 = Flow Solution Loads 2
Operator 2 = boundary sum
End

Boundary Condition 3
Name = no_slip
...
Save Scalars = Logical True
End
```

Benchmark information

The following example shows how benchmark information from the computation of some potential equation is gathered to one file. Different runs will append the results to the same file and in parallel runs the degrees of freedom, number of partitions and consumed cpu time will be automatically gathered.

```
Solver 3
Exec Solver = After all
Equation = SaveScalars
Procedure = "SaveData" "SaveScalars"
Filename = timing.dat
File Append = Logical True
Variable 1 = Potential
Operator 1 = dofs
Operator 2 = partitions
Operator 3 = cpu time
Parallel Reduce = Logical True
End
```

Model 49

Saving data along lines to a file

Module name: SaveData

Module subroutines: SaveLine

Subroutine authors: Peter Råback, Ville Savolainen

Document authors: Peter Råback

Document created: 3.10.2002

Document updated: 8.4.2017

49.1 Introduction

The subroutine saves lines that pass through higher dimensional computational meshes. The data is saved in simple matrix format thereby allowing the easy postprocessing of data by MatLab, Excel or any other program that can read ASCII data. In addition to the number values also an additional file with the suffix `.name` is saved. It tells what variables are at each column.

49.2 Theory

One mildly theoretical problem in saving data comes from the fact that the data should be saved in lines that were not a priori defined. If there are relatively few points the dummy algorithm where each element is checked for including the node may be used. For the lines, however, this algorithm might become quite expensive as there may be many points that constitute the line and faster choices are needed.

Therefore we only look for intersections of element faces and the lines. Each element face is divided into triangles. The triangle has points \vec{e}_1 , \vec{e}_2 and \vec{e}_3 . The line is drawn between points \vec{r}_1 and \vec{r}_2 . Therefore the line goes through the point only if

$$\vec{r}_1 + a(\vec{r}_2 - \vec{r}_1) = \vec{e}_1 + b(\vec{e}_2 - \vec{e}_1) + c(\vec{e}_3 - \vec{e}_1) \quad (49.1)$$

has a solution for which $0 \leq a, b, c \leq 1$. This results to a matrix equation

$$\begin{pmatrix} r_{2x} - r_{1x} & e_{1x} - e_{2x} & e_{1x} - e_{3x} \\ r_{2y} - r_{1y} & e_{1y} - e_{2y} & e_{1y} - e_{3y} \\ r_{2z} - r_{1z} & e_{1z} - e_{2z} & e_{1z} - e_{3z} \end{pmatrix} \begin{pmatrix} a \\ b \\ c \end{pmatrix} = \begin{pmatrix} e_{1x} - r_{1x} \\ e_{1y} - r_{1y} \\ e_{1z} - r_{1z} \end{pmatrix} \quad (49.2)$$

which may be easily solved with standard methods linear algebra. Because the face element is a triangle there is an additional condition that $b + c \leq 1$.

49.3 Keywords

Solver `solver id`

Procedure File "SaveData" "SaveLine"

Filename String

Name of the file where results are to be saved, the default is `sides.dat`.

Output Directory String

Name of the directory where results are to be saved, relative to the case directory. By default the results are saved in the case directory.

File Append Logical

If the results from consecutive rounds should be appended to the file this flag should be set to True. The default is False.

Save Axis Logical

Save all the principal axis. Also keywords `Save Axis i` exist, where $i=1,2,3$ defines the axis.

Polyline Coordinates (n, DIM) Real

Save the line consisting of line segments defined by two points ($n = 2$). There can be more than one set of points ($n = 2, 4, 6, \dots$) but as a line segment is defined by two points there must be an even number of points.

Polyline Divisions ($n/2, DIM$) Integer

The user may give the number of divisions for each polyline. This allows also the proper saving of discontinuous data. The size of this vector should be such that it is compatible with the number of lines.

Save Isocurves Logical

Saves isocurves defined on 2D meshes.

Isosurface Variable i String

The variable which isocurve to save, $i = 1, 2, \dots$. This must be a scalar variable.

Isosurface Value i Real

A constant value that defines the value of the isosurface variable at the isocurve to be saved. Note that for the same variable there may be several values, each with a different keyword.

Variable i String

By default `SaveLine` saves all the active variables. However, it is possible to save only a specified list of variables given by this keyword where $i=1,2,3,\dots$. This may be particularly useful if one wants to save a table of linear dependence, for example Temperature along x -direction, to be used as a boundary condition in consecutive Elmer runs with a different mesh.

Save Flux Logical

Saves a flux resulting from a gradient of a field by the model $h = -\kappa \partial T / \partial n$. This may only be applied to existing boundaries, not lines defined by points.

Flux Variable String

The name of the field variable (default T is Temperature).

Flux Coefficient String

The diffusion constant (by default κ is Heat Conductivity)

Save Mask String

By default `SaveLine` saves only the values that are on boundary marked with `Save Line` flag. If the user wants several instances of the `SaveLine` subroutine, for saving different boundaries to different files, the mask name may be defined by this keyword. The correspondingly one should use the same flag in the `Boundary Condition` and `Body` section.

Boundary Condition `bc id`

Save Line Logical

The flag activates the saving of the boundary condition as a line. The subroutine tries to save the finite-element lines as a chain of points to enable nice preprocessing with MatLab or similar tools. The flux may only be saved on lines defined by boundary conditions.

49.4 Examples

In the following examples it is assumed that the 1st solver is the primary solver and the `SaveLine` solver gets index two.

The following example shows how to save a line that extends from point (0,0,0) to point (1,2,3) in 3D mesh after the whole simulation has ended using 100 divisions for both line segments.

```
Solver 2
  Exec Solver = After Simulation
  Equation = SaveLine
  Procedure = "SaveData" "SaveLine"
  Filename = "line.dat"
  Polyline Coordinates(2,3) = 0.0 0.0 0.0 1.0 2.0 3.0
  Polyline Divisions(2) = 100 100
End
```

Model 50

Saving material parameters and boundary conditions

Module name: SaveData

Module subroutines: SaveMaterials, SaveBoundaryValues

Module authors: Thomas Zwinger

Document authors: Thomas Zwinger, Peter Råback

Document created: 3.10.2002

Document updated: 4.4.2011

50.1 Introduction

These subroutines creates fields from material parameter entries or boundary conditions that normally cannot accessed as a field, as they are expressions that is evaluated when needed. The `SaveMaterials` may be used to create additional field variables from the material parameters. A similar procedure `SaveBoundaryValues` stores parameters defined on boundaries as variables for the whole mesh. This can be of help if a boundary condition that is not directly accessible from the variables (like a normal component of a vector field) should be evaluated in the post-processing step.

50.2 Keywords

Keywords of subroutine `SaveMaterials`

Solver `solver id`

```
Procedure File "SaveData" "SaveMaterials"
```

```
Parameter i String
```

The user may choose a number of parameters ($i=1, \dots, 99$) which will be save as variables. This may be particularly handy if one wants to visualize how the parameters depend on the position over the domain. Values in bodies with the assigned material list not containing the keyword of the parameter are set to zero by default.

Keywords of subroutine `SaveBoundaryValues`

Solver `solver id`

```
Procedure File "SaveData" "SaveBoundaryValues"
```


Variable String -nooutput dummyvar
a dummy variable for the solver that does not show up

Variable DOFs Integer 1

Parameter i String

The user may choose a number of parameters ($i=1,\dots,99$) which will be save as variables. These parameters will then be stored as variables with the values assigned as they were found on the specific boundary. Bulk values and values on boundaries with the parameter not being defined are set to zero by default.

Body Force Parameters Integer

The user may also save parameters given in body force section by giving the number of these parameters by this keyword. Note that the body force parameters must be first in the list followed by the material parameters. The default is zero.

Model 51

Result output in different formats

Module name: ResultOutputSolve

Module subroutines: ResultOutputSolver

Module authors: Peter Råback, Erik Edelman, Mikko Lyly

Document authors: Peter Råback

Document created: 11.12.2006

Document edited: 22.9.2014

51.1 Introduction

This subroutine is intended for saving data in other than the native format of Elmer – ElmerPost. The reason for using another postprocessing tool might be that some feature is missing in ElmerPost, or that the user is more acquainted with some other visualization software. Currently supported formats include GiD, Gmsh, VTK legacy, XML coded VTK file bearing the suffix VTU and Open DX.

The recommended 3rd party visualization tool of Elmer results is Paraview and the corresponding format for it is the Vtu format. The old VTK format is not recommended.

51.2 Keywords

Solver solver id

Equation String "ResultOutput"

The name of the equation. This is actually not much needed since there are no degrees of freedom associated with this solver.

Procedure File "ResultOutputSolve" "ResultOutputSolver"

The name of the file and subroutine.

Output File Name File

Specifies the name of the output file.

Output Format String

This keyword the output format of choice. The choices are gid, gmsh, vtk, vtu, and dx.

Gid Format Logical

Gmsh Format Logical

Vtk Format Logical

Vtu Format Logical

Dx Format Logical

The user may also use the above logical keywords to set which of the formats is saved. This

has more flexibility in that there may be several formats that are saved simultaneously where the Output Format keyword may only be used to activate one solution type.

Binary Output Logical

For Vtu format (no other format) the data may also be written in binary format which is significantly more compact than the ascii format. Default is True.

Ascii Output Logical

As the default format for Vtu is binary this keyword is more natural for enforcing ascii format. The default is False.

Single Precision Logical

The floating numbers in Vtu format may be saved either in single (32 bits) or double (64 bits) precision. The single precision saves some disk space. Default is False.

Eigen Analysis Logical

For GiD and Gmsh output format activates the eigenmode writing, and in Vtu format makes the numbering of the files follow the eigenmodes. Vtu format will still save the eigenmodes without this keyword but this will be done in the same file just altering the name of the field.

Number Of EigenModes Integer

Maximum number of modes, when supported. Default is that all eigenmodes are saved.

Active EigenModes Integer

List of active eigenmodes to be saved. Supported only in the Vtu format. By default modes are saved in order (1,2,...).

The following keywords related only to the GiD, Vtu and Gmsh formats. In the other formats all available degrees of freedom are saved. Also in Vtu format all dofs are saved if none of the list is given.

Scalar Field i String

The scalar fields to be saved, for example Pressure. Note that the fields must be numbered continuously starting from one.

Vector Field i String

The vector fields to be saved, for example Velocity

Tensor Field i String

The tensor fields to be saved. The rank of tensor fields should be 3 in 2D and 6 in 3D.

Sometimes when the variables need to be explicitly listed it may be difficult to know what the actual available variables are. For this purpose there is the following keyword.

Show Variables Logical

Show all the different variables on output as a list. Default is False.

In the Vtu output format the user may use several various masking operations to choose the elements to be saved. These cannot be used with other formats. Also the user may choose whether to save elemental or nodal fields, if applicable.

Save Geometry Ids Logical

Save the index of geometric entities i.e. of the bodies and boundaries. The body index will be saved as such whereas the boundary index will get a offset of 100 (or always 10 times larger if not larger than largest body index).

Default Body Id Integer

This may be used to modify the default geometry id for the bodies when saving the geometry ids in VTU format.

Default BC Id Integer

This may be used to modify the default geometry id for the boundary conditions when saving the geometry ids in VTU format.

BC Id Offset Integer

This may be used to modify the offset from the default one when saving the geometry ids in VTU format.

Save Elemental Fields Logical

There may be some elemental fields present, for example if they have been computed with the Discontinuous Galerkin method. Then the elementwise nodal information is average to elemental field. Elemental fields can present discontinuities better than the nodal fields, whereas the nodal fields can represent smooth fields better.

Save Nodal Fields Logical

Save the computed nodal fields. Most fields are nodal only.

Save Linear Elements Logical

For higher order elements the user has the option to save the data still using linear basis. This effectively saves only values at the corner nodes of each element. Often the derived data from quadratic elements is just of linear accuracy and therefore there is no real benefit in treating the actual quadratic data.

Discontinuous Galerkin Logical

When dealing with elemental fields there is the option to save the field as discontinuous such that each instance of a node is recreated. This may make the resulting files huge but may still be the desired option when dealing with discontinuous fields.

Discontinuous Bodies Logical

When dealing with elemental fields one can also create a minimal discontinuous set of nodes such that the discontinuous information is averaged within shared nodes body-wise. Often the discontinuity is present only over different bodies so this strategy maintains the essential discontinuous while having only a minor effect on the file size.

Vtu Time Collection Logical

Save a time collection file with a .pvd suffix that includes the timestamps for each file in a transient simulation.

Skip Halo Elements Logical

If halo element are used in the parallel computation this flag can be used to suppress the saving of these elements. They are basically redundant so this flag could well be set to true for typical use.

Save Halo Elements Only Logical

If halo elements are used then this flag can be used to save only the halo elements. This would probably have mainly uses in debugging or demonstrating the halo elements.

Save Bulk Only Logical

Save the bulk elements only.

Save Boundaries Only Logical

Save the boundary elements only. This could be useful if one is interested of the results only at the boundaries. This flag would also often save considerable amount of disk space.

Mask Variable String

The user may give a variable for masking. If this flag is given then only the elements where the permutation vector associated with this variable is positive for all nodes are saved.

Mask Name String

The user may give a logical name of the mask. This mask will then be checked for in body force and boundary condition lists and the active elements are determined on-the-fly.

Mask Condition String

The user may give a real valued condition for the mask. This mask condition will be check in body for and boundary condition lists and the results will be saved only where the value of the mask condition is positive. This could easily be used to save results only within a sphere, for example.

```
Body    body id
```

```
    Geometry Id Integer
```

May be used to remap the default body index to a new value for VTU output.

```
Boundary Condition bc id
```

```
    Geometry Id Integer
```

May be used to remap the default boundary condition index to a new value for VTU output.

If the user only wants a basic functionality of `VtuOutputSolver` it is possible to let the system create automatically an instance of the solver imply by adding a `Post File` with suffix `.vtu`.

```
Simulation
```

```
    Post File File [filename.vtu]
```

Name of the file where the unstructured XML based VTK results are to be saved.

```
    vtu: Keyword Type
```

Any keyword with the suffix `vtu:` is passed to the solver instance without the suffix.

By default the solver is executed after saving. With the help of the namespace it is possible to include any keyword (with the keyword type given) also in the `Simulation` section. At some point the section will get crowded making it justified to create a separate solver instance.

51.3 Example

This example saves the results of a computation in binary unstructured VTK XML format following the `Output Intervals` definition in the `Simulation` section. The variables to be saved are hand-selected.

```
Solver 3
```

```
    Exec Solver = after saving
```

```
    Equation = "result output"
```

```
    Procedure = "ResultOutputSolve" "ResultOutputSolver"
```

```
    Output File Name = "case"
```

```
    Vtu Format = Logical True
```

```
    Binary Output = Logical True      ! binary format is the default
```

```
    Single Precision = Logical True   ! double precision is the default
```

```
! In the Vtu format all fields are saved if the user does not list them explicitly.
```

```
    Scalar Field 1 = String Temperature
```

```
    Scalar Field 2 = String Pressure
```

```
    Vector Field 1 = String Velocity
```

```
End
```

Model 52

Saving data on uniform cartesian grid

Module name: SaveGridData
Module subroutines: SaveGridData
Module authors: Peter Råback
Document authors: Peter Råback
Document created: 2.2.2011
Document edited: 20.7.2011

52.1 Introduction

This subroutine is intended for saving data in a uniform grid. One possible use of the feature is to combine a boundary representation and a glyph representation (velocity vectors) in the same visualization. This routine would then save the data for the glyphs. The routine may also be used so that the resolution in two other directions is set to a high value which will then effectively create uniform line plots.

The algorithm goes through all the elements and checks whether the element could include some of the uniform grid nodes. If so then these are tested for. The algorithm should scale linearly with mesh size. Optionally one may check for duplicates to eliminate the same nodes being repeatedly saved. Currently no particular order is guaranteed for the nodes.

The routine works currently only in VTU, VTI and ascii table formats, but other format may be possible in the future.

52.2 Keywords

Solver `solver id`

Equation `String "SaveGridData"`

The name of the equation. This is actually not much needed since there are no degrees of freedom associated with this solver.

Procedure `File "SaveGridData" "SaveGridData"`

The name of the file and subroutine.

Filename Prefix `File`

Specifies the name of the output file. A appropriate suffix is added to the given name.

Output Format `String`

This keyword the output format of choice. The choices are `vtu`, `vti` and `table`.

Vtu Format `Logical`

Vti Format `Logical`

Table Format Logical

The user may also use the above logical keywords to set which of the formats is saved. This has more flexibility in that there may be several formats that are saved simultaneously where the **Output Format** keyword may only be used to activate one solution type.

Fileindex Offset Integer

By default the files are numbered starting from one. However, for some restarted simulations the offset may be defined to be something else. The default is zero.

Check for Duplicates Logical

This flag activates the checking of duplicates. This is usually advisable but may for peculiar geometries require a large logical table and is therefore not defaulted to be true.

Grid dx Real

The grid size in direction x (i.e. **Coordinate 1**). Similarly we have **Grid dy** and **Grid dz**, if applicable. If the density in directions y or z is not defined, it is assumed to be the same as for x .

Grid nx Integer

The number of cells in direction x . This is only required if the previous keyword is not given. Then the resolution is defined by the size of the bounding box. Similarly we have **Grid ny** and **Grid nz**,

Grid Origin i Real

The mesh is by default located so that there is a node at $(0, 0, 0)$. If the origin should not coincide here then this keyword may be used to transform the origin ($i = 1, 2, 3$).

Min Coordinate i Real

The bounding box may be limited by this keyword ($i = 1, 2, 3$). If not given the minimum values of the mesh are used.

Max Coordinate i Real

The bounding box may be limited by this keyword ($i = 1, 2, 3$). If not given the maximum values of the mesh are used.

Scalar Field i String

The scalar fields to be saved, for example **Pressure**. Note that the fields must be numbered continuously starting from one. If no fields are given then an attempt is made to save all the relevant fields.

Vector Field i String

The vector fields to be saved, for example **Velocity**

Mask Name String [MyMask]

The user may provide a mask that is used to determine the active elements. If the elements are lower dimensional then it is assumed that the last coordinate is eliminated from the gridded data. So if the full mesh is 3D and a mask is given for a boundary only then the data is saved on x-y plane.

Filename Timestep Numbering Logical

Use this keyword in transient case to number the files by the timesteps. Applies only to the table format.

Filename Particle Numbering Logical

Use this keyword in transient case to number the files by particles indexes. Applies only to the table format.

Boundary Condition bc id**MyMask** Logical True

The mask define in the solver section may be set True in the BC or Body Force section.

52.3 Example

The following example saves the results in the VTK XML image data format at the end of simulation. A mesh parameter $h = 0.1$ is used for every direction and the lower left corner is taken as the base of the finite uniform mesh.

```
Solver 5
  Exec Solver = after all
  Equation = SaveGrid
  Procedure = "SaveGridData" "SaveGridData"

  Grid dx = Real 0.1
  Grid Origin At Corner = Logical True
  Check For Duplicates = Logical True
  Binary Output = Logical True
  Single Precision = Logical True
  Filename Prefix = String glyphs
  Vti Format = Logical True
End
```


Model 53

Isosurface extraction for reduced output

Module name: Isosurface

Module subroutines: IsosurfaceSolver

Module authors: Juha Ruokolainen, Peter Råback

Document authors: Peter Råback

Document created: 15.2.2011

Document edited: 15.2.2011

53.1 Introduction

This subroutine extract isosurfaces from 3D meshes or isolines from 2D meshes. The intended use of the routine is in heavy simulations where the standard output could result to an I/O bottle-neck. If the desired isosurface is known in advance then this can be used to reduce the amount of data to be written. The solver does not itself write the data. It is expected that there is some external strategy for writing the data. It could, for example, be the `ResultOutputSolver` for VTK XML output.

53.2 Keywords

`Solver` `solver id`

`Equation` `String` `"Isosurface"`

The name of the equation. This is actually not much needed since there are no degrees of freedom associated with this solver.

`Procedure` `File` `"Isosurface"` `"IsosurfaceSolver"`

The name of the file and subroutine.

`Output Directory` `File`

Specifies the name of the output directory. Output file name will be determined in the normal manner.

`Isosurface Variable` `String`

The name of the variable for which the isosurface is determined. This must be a scalar variable.

`Isosurface Values` `Real`

The isosurface values used to extract the surfaces. If this is a vector then a number of isosurfaces will be defined.

`Isosurface Values` `Real`

The isosurface value used to extract the surfaces. For constant values this is the scalar variant of the previous keyword. However, this may also have a functional dependency on some scalar variable, such as time, for example.

```
Variable i String
```

Name of the variable to be outputted on the isosurface (or isoline), $i=1, 2, 3, \dots$

53.3 Example

This solver extracts the 1D isolines from 2D temperature field and stores them to a mesh called `isosurf` at the three given isosurface values. If the original mesh would be 3D then the resulting isosurface mesh would be 2D. Note that if saving of results is desired then this solver should be performed prior to the saving of results.

```
Solver 8
  Exec Solver = after all
  Equation = "isoline"
  Procedure = "Isosurface" "IsosurfaceSolver"
  IsoSurface Variable = Temperature
  IsoSurface Values(3) = 0.25 0.5 0.75

  Output Directory = isoline

  Variable 1 = Temperature
  Variable 2 = stream
  Variable 3 = vorticity
End
```

Model 54

Coupling Elmer with OpenFOAM via file IO

Module name: Elmer2OpenFoamIO,OpenFOAM2ElmerIO

Module subroutines: Elmer2OpenFoamIO,OpenFOAM2ElmerFit

Module authors: Peter Råback

Document authors: Peter Råback

Document created: 24.3.2017

Document edited: 31.5.2018

54.1 Introduction

These solver modules provide the possibility to transfer field values from Elmer into OpenFOAM and back. The data transfer is done via file IO and has therefore some limitations regarding speed of operation. However, the approach also has some flexibility since it does not set any additional constraints to how Elmer and OpenFOAM have been compiled.

The solvers assume a working Elmer case and OpenFOAM case directory. The solvers assume that the OpenFOAM directory 0 includes the cell centers computed with `writeCellCenters` in file C. For a multiblock case the mesh directory will include subdirectories and then the cell centers should be available at least in one subdirectory. Note that the Elmer field will only be mapped if the cell center file C is provided.

The Elmer2OpenFoamIO solver first reads the cell centers, then creates a temporal mesh structure from them, and interpolates the desired field using the standard interpolation routines within Elmer. The interpolation uses octree search tree and therefore it should be rather fast – in serial. The interpolated field is written into the same directory where cell centers C were found. The format is a one that can be used to initialize object from class `volScalarField`.

The coupling from OpenFOAM to Elmer is performed with the OpenFOAM2ElmerFit routine. It again reads the cell centers from files C. The solution with a given name is not fetched from directory 0 but rather from the timestep given by the user. The resulting values are not interpolated using the basis function since there is no OpenFOAM mesh available at Elmer. Rather a procedure similar to `DataToFieldSolver` is used. There diffusivity may be given as a regularization parameter. A good choice may be the physical diffusivity related to the solver. I.e. if you're reading in temperatures they may be regularized by heat conductivity.

In case the performance of the routines is not satisfactory or one needs parallel operation then the MPI coupler `EOF_library` mainly written by Juris Vencels could be a better choice for the problem.

54.2 Keywords for module Elmer2OpenFoamIO

Solver `solver id`

Equation String [Elmer2OpenFoamIO]
The name of the equation.

Procedure File "Elmer2OpenFoamIO" "Elmer2OpenFoamWrite"
The name of the procedure for writing Elmer fields to be read by OpenFOAM.

Filename File
Full name of the target file (with the suffix).

Target Variable String
Name of the Elmer field to be mapped for OpenFOAM. Currently only one field is supported.

OpenFoam Directory File
Name of the OpenFOAM directory that includes the whole OpenFOAM case tree.

OpenFoam File File
Name of the OpenFOAM file that will include the exported field from Elmer interpolated to the cell centers.

OpenFoam Mesh i File
If the OpenFOAM mesh directory cannot be retrieved automatically for some reason the user may define the OpenFOAM meshes to be treated. This could be the case in Windows where the system commands might not be available.

54.3 Keywords for module OpenFoam2ElmerIO

Solver solver id

Equation String [OpenFoam2ElmerIO]
The name of the equation.

Procedure File "penFoam2ElmerIO" "OpenFoam2ElmerFit"
The name of the procedure for importing fields from OpenFOAM to Elmer.

OpenFoam Directory File
Name of the OpenFOAM directory that includes the whole OpenFOAM case tree.

OpenFoam Mesh i File
If the OpenFOAM mesh directory cannot be retrieved automatically for some reason the user may define the OpenFOAM meshes to be treated. This could be the case in Windows where the system commands might not be available.

OpenFoam Field String
Name of the OpenFOAM field to be read from the directory tree. Currently only one scalar field is supported. Typical files are T (temparature) and p (pressure).

OpenFoam Timestep Integer
Name timestep number to define the subdirectory where the field is read from. Zero refers to the initial state.

Fit Coefficient Real
This constant gives the factor that is used to weight the data when performing the fitting. This has a relevance only with respect to the diffusivity used for regularization. Default value is one.

Diffusion Coefficient Real
This constant gives the diffusivity used for regularization.

Diffusivity Name String
The user may give an existing material parameter for the diffusivity. Then this is the name which is searched in Material section.

Passive OpenFOAM Coordinate Integer 3
In case the initial file is 3D and we want to interpolate the results into a 2D mesh we may give the paassive coordinate index that will be used in dimensional reduction. If not given, 3D interpolation will be assumed.

54.4 Example

This example is used to map the field `joule heating` from Elmer to the OpenFOAM initialization file `fieldSolidHS.dat` to be used in temperature distribution computations.

```
Solver 2
  Equation = ElmerToOpenFOAM
  Procedure = "Elmer2OpenFoamIO" "Elmer2OpenFoamWrite"

! The variable to be mapped
  Target Variable = joule heating

! The OpenFOAM project directory containing the mesh etc.
  OpenFOAM Directory = FILE "ofdir"

! The file to write the OpenFOAM sources to.
  OpenFOAM File = File "fieldSolidHS.dat"
End
```

Model 55

Reload Existing Simulation Results

Module name: ReloadData

Module subroutines: ReloadSolution

Module authors: Antti Pursula

Document authors: Antti Pursula

Document created: August 9th 2007

55.1 Introduction

This subroutine is intended for repeated loading of existing results during simulation. An example of a typical application is to use previously computed fluid flow as a convection field for the transfer of a passive scalar variable. The module is implemented as a dummy solver which is defined in the command file just as the 'normal' solvers.

This module offers extended features compared to the `Restart File` option in the `Simulation` section. The module reads a new solution step from the solution file on each timestep, whereas the restart file option reads only the initial state for a simulation.

The module reads in all the available variables from the solution file. The solution file should be in the mesh directory. If the simulation takes more than a single steady state iteration per time step it is advisable to use `Exec Solver = Before Timestep` for this module.

55.2 Keywords

`Solver` `solver id`

`Equation` `String` "Reload Data"

The name of the equation. This is actually not much needed since there are no degrees of freedom associated with this solver.

`Procedure` `File` "ReloadData" "ReloadSolution"

The name of the file and subroutine.

`Reload Solution File` `String` "flow-data.dat"

The name of the old solution data file, eg. flow-data.dat

`Reload Starting Position` `Integer`

The index of the timestep where to start reading. If the keyword is not given the reading is started from the first step in the file, or from the beginning of the reload range, if specified.

`Reload Range Minimum` `Integer`

`Reload Range Maximum` `Integer`

The beginning and the end of the reading range. The timesteps on the range are read in cyclically if the current simulation has more timesteps than what there are on the range.

Reload Reading Intervals Integer

Defines the interval for reading in old results, defaults to 1. An integer i larger than 1 defines that results are read in only on every i th timestep. However, consecutive steps are read in regardless of the value of i .

Continuous Reading Logical True

When set to `True` the reload solution file is kept open also between the timesteps. However, when reading is not started at the first solution step, or when the old solution is read in cyclically, it is advisable to switch this feature off. Defining `False` will slow down reading especially from large ASCII files.

Model 56

Runtime Control of the Input Data

Module name: ReloadInput

Module subroutines: ReloadInput

Module authors: Juha Ruokolainen

Document authors: Peter Råback

Document created: February 5th 2003

56.1 Introduction

This subroutine is intended for cases where the user wants to have run-time control over the solution. The control is obtained by reloading the command file (.sif-file) during the solution. This is done with an additional solver that is called similarly as any other solver during the solution process.

The most likely usage of the solver is in cases where the user realizes during the solution process that some parameters were not optimally chosen. For example, the convergence criteria may have been set too tight for optimal performance. Then the user may set looser criteria by editing the command file during the computation. Once the new value is read the solver will apply the new criteria thereafter.

56.2 Limitations

The solver should not be used for things that need allocation. For example, the number of solvers or boundaries may not change. Also the computational mesh must remain the same.

56.3 Keywords

Solver solver id

Equation String "Reload"

The name of the equation. This is actually not much needed since there are no degrees of freedom associated with this solver.

Procedure File "ReloadInput" "ReloadInput"

The name of the file and subroutine.

Model 57

Reading NetCDF data into FE mesh

Module name: GridDataReader
Module subroutines: GridDataReader
Module authors: Peter Råback, Vili Forsell
Module status: Alpha
Document authors: Peter Råback
Document created: 7.12.2011
Document edited: 22.12.2011

57.1 Introduction

This solver provides the possibility to read in data in uniform grid into Elmer mesh. The supported format includes currently only NetCDF.

Currently following features are supported:

- Bilinear (in 2D) or trilinear (in 3D) interpolation
- Possibility to have a multiplier and offset in the target field
- Linear interpolation in time
- Multiple possibilities to define the time instant or index

The module requires the netcdf library. Because of this, it has been isolated from the main build system. The source code for XdmfWriter can be found from the source tree in `misc/netcdf2` and it should be compiled by the user as follows:

```
elmerf90 -I$HDF5/include -L$HDF5/lib -o GridDataReader.so GridDataReader.f90 -lnetcdff -lnetcdf
```

The environment variable `$NETCDF` defines the installation directory for the NetCDF-library.

57.2 Theory

In structured data the finding of grid cell where a certain node belongs to is trivial. Assume that the grid is defined by coordinates a_i and b_i so that any coordinate x_i is presented by

$$x_i = a_i n_i + b_i \quad (57.1)$$

where $n_i \in [1, N_i]$ and $i \in [1, \text{DIM}]$. Now the finite element node with coordinate \hat{x}_i may be interpolated using the cell with indexes $m_i = \text{ceiling}((\hat{x}_i - b_i)/a_i)$ and $m_i + 1$ using linear interpolation with weighing factor $q_i = m_i - (\hat{x}_i - b_i)/a_i$.

$$\hat{f} = q_i f_{m_i} + (1 - q_i) f_{m_i+1}. \quad (57.2)$$

This generalizes into multiple dimensions using recursion. For example, in 2D the bilinear interpolation reads

$$\hat{f}(x_1, x_2) = q_1 q_2 f_{11} + (1 - q_1) q_2 f_{21} + q_1 (1 - q_2) f_{12} + (1 - q_1) (1 - q_2) f_{22}. \quad (57.3)$$

The current implementation provides bilinear and trilinear interpolation in space, and linear interpolation in time.

The algorithmic complexity of the reader and interpolation routine is linear in time. However, the current implementation assumes that each partition do their own interpolation routines which means that there may be a large number of files open in parallel runs introducing potential bottle-necks.

57.3 Keywords

Solver solver id

Equation String [GridDataReader]

The name of the equation.

Procedure File "GridDataReader" "GridDataReader"

The name of the procedure.

Filename File

Full aame of the target file (with the suffix).

X Name String

Name of the 1st coordinate.

Y Name String

Name of the 2nd coordinate.

Z Name String

Name of the 3rd coordinate.

Time Name String

Name of the time coordinate.

X Epsilon A

ccuracy of the 1st coordinate assumed in interpolation. Default is machine epsilon.

Y Epsilon A

ccuracy of the 2nd coordinate assumed in interpolation. Default is X Epsilon.

Z Epsilon A

ccuracy of the 3rd coordinate assumed in interpolation. Default is X Epsilon.

Time Epsilon A

ccuracy of time assumed in interpolation. Default is machine epsilon.

Time Point Real

Value of time used in this calling. If this is not found the Elmer time will be used.

Time Offset Real

Offset used to add to the Elmer time.

Time Multiplier Real

Coefficient used to multiply the Elmer time.

Is Time Index Real

If this flag is turned on then the time instance given with the previous keywords will be understood as being the index of time in the NetCDF file (starting from 1) rather than actual time.

Is Time Counter Logical

If this flag is turned on, the time index of the NetCDF file will be increased by one each time the routine is called. This makes it ideal for looping over each timestep.

Coordinate Transformation String

Optional coordinate transformation that is applied on the Elmer coordinate prior to finding its value on the grid.

Enable Scaling Logical

If this flag is turned on the bounding box of Elmer mesh will be made compatible with that of the grid.

Variable i String

Variable of the NetCDF file. There is no upper limit to the number of variables.

Target Variable i String

Optional name for the target variable. If this is not given then the Variable i value will be used for the FE variable as well. If the target variable is not present it will be created the first time it is needed. Also, if different variables that follow each other have the same target variable these will be summed up. With the multiplication and offset features this makes it possible to derive new variables as a linear combination of any fields given in the NetCDF file.

Interpolation Offset Real

A constant that will be added to the value of the interpolated field. If there are many fields with different offsets use Interpolation Offset i instead.

Interpolation Multiplier Real

A constant that will be used to multiply the interpolated field. The default value is one. If there are many fields with different offsets use Interpolation Multiplier i instead.

Model 58

Filtering Time-Series Data

Module name: FilterTimeSeries

Module subroutines: FilterTimeSeries

Module authors: Peter Råback

Document authors: Peter Råback

Document created: 13.2.2008

Document updated: 13.2.2008

58.1 Introduction

The module includes auxiliary utilities for filtering time-series data. Supported filters include various averaging possibilities and Fourier series, for example. The solver does not introduce any new physics. However, it may be useful in analyzing time-dependent data to be used in conjunction with time-harmonic models, or in studying phenomena with different timescales (turbulence).

58.2 Theory

Mean of a function

The solver is built so that an estimate for the filtered data may be obtained at all times i.e. the normalizing is done after each timestep. As an example let's consider taking a simple mean over a period of time. The starting point is the time averaged mean,

$$\langle f \rangle_T = \frac{1}{T} \int_0^T f(t) dt. \quad (58.1)$$

Its discrete counterpart assuming piecewise constant integration is

$$\langle f \rangle_n = \frac{1}{T_n} \sum_i^n f_i dt_i, \quad (58.2)$$

where $T_n = \sum dt_i$. Now this may be presented inductively as

$$\langle f \rangle_n = \frac{T_{n-1} \langle f \rangle_{n-1} + f_n dt_n}{T_{n-1} + dt_n} \quad (58.3)$$

$$T_n = T_{n-1} + dt_n. \quad (58.4)$$

Weighted mean

It's also possible to take a weighted mean with a user defined function $g(t)$ depending on time only. Then similarly,

$$\langle fg \rangle_n = \frac{T_{n-1} \langle fg \rangle_{n-1} + f_n g_n dt_n}{T_{n-1} + dt_n}. \quad (58.5)$$

Fourier series

Using the weighted mean as starting point its possible to present the solution in terms of sine and cosine series. In order to obtain normalized Fourier series components the function g is internally replaced by sine and cosine functions defined as $2 \sin(2\pi kwt)$ and $2 \cos(2\pi kwt)$, where k is the degree of the term, and w is the user defined frequency. After each full cycle the inner product then includes the Fourier coefficients and the transient solution may hence be approximated by

$$f \approx \sum_{k=1}^{m_s} s_k \sin(2\pi kwt) + \sum_{k=1}^{m_c} c_k \cos(2\pi kwt), \quad (58.6)$$

where m_s and m_c are maximum degrees defined by the user.

Continuous average

Sometimes it may be useful that the new solution is given a relatively higher weight than the old solution. This is achieved by relaxing the weight (elapsed time) related to the old solution by

$$T_{n-1} := T_{n-1} \exp(-dt_n/\tau), \quad (58.7)$$

where τ is the time scale when decay to fraction $1/e$ is desired. If the decay time is short compared to the overall simulation time this provides a continuous mean that represents only the recent results. The fraction of the last timestep in solution will always be dt/τ .

Computing variances

It is not possible to compute the variance directly with one sweep as computing the variance from the functional values requires the knowledge of the mean. However, computing the mean of the square of the solution enables that the variance is computed a posteriori since the following holds for any field variable,

$$\sigma^2 = \langle (f - \langle f \rangle)^2 \rangle = \langle f^2 \rangle - \langle f \rangle^2. \quad (58.8)$$

58.3 Keywords

Solver `solver id`

Procedure `File "FilterTimeSeries" "FilterTimeSeries"`

Variable `i String`

The names of the variables to be filtered. There can in principle be up to 99 variables. Note that the keywords with the same `i` form a set which define one filtering. If the `Variable` is not redefined the previously defined variable with a lower `i` is used.

Operator `i String`

Normally the variable is treated as its plain value. There are however different options for using the field value in a modified manner. These include `length` (L2 norm), `abs`, and `square`.

Start Time `i Real`

The start time for performing the integration. Note that for Fourier series this is used to reset the zero time i.e. $t := t - t_0$.

Stop Time *i* Real

The stop time for performing the integration.

Start Timestep *i* Integer

Sometimes its unpractical to compute the start time. For example, the start of the simulation could include a starting strategy with a number of timesteps. Then the number of timesteps that starts the averaging may be given by this keyword. Note that this keyword also activates timestep-insensitive averaging.

Stop Timestep *i* Integer

The timestep number that ends the averaging.

Start Cycle *i* Real

Alternative way to give the start time for sine and cosine series. The start time is the inverse of this.

Stop Cycle *i* Real

Alternative way to give the stop time for sine and cosine series.

Start Real Time Real

Start after given real wall-clock-time, rather than physical simulation time.

Start Real Time Fraction Real

Relative way of given start time when the Real Time Max keyword in Simulation block is given.

Reset Interval *i* Real

The time interval at which the computation of a mean is reinitialized.

Decay Time *i* Real

The decay time τ in computing continuous means.

Decay Timestep *i* Real

The number of timestep needed to perform averaging. If the timestep given is N then the decay of the previous timesteps is done with $\exp(-1/N)$. Note that this keyword also activates timestep-insensitive averaging.

Time Filter *i* Real

The function $g(t)$ that may be used in computing the mean.

Sine Series *i* Integer

The number of terms in the sine series. Note that its possible to make a Fourier series only if the target variable is a scalar. Its also possible to have only one sine or cosine series at a time.

Cosine Series *i* Integer

The number of terms in the cosine series.

Frequency *i* Integer

If using cosine or sine series the frequency must be given.

Model 59

Data to field solver

Module name: DataToFieldSolver
Module subroutines: DataToFieldSolver
Module authors: Peter Råback
Module status: Alpha
Document authors: Peter Råback
Document created: 4.8.2011
Document edited: 7.3.2013

59.1 Introduction

This subroutine may be used fit data to a continuous field. In principle just a simple identity equation with some diffusion for regularization is solved. The origin of the data could be from a particle trace of a Monte-Carlo simulation, measurement data read from an external file etc.

59.2 Theory

Galerkin method minimizes the L2 norm of the solution and is therefore a good choice also for problems where a regular field must be fitted into data. Imagine a case where we have an initial field \hat{f} and want to a field f on it. Then we need to solve an equation $f = \hat{f}$ with the Galerkin method. Now if the initial data is noisy the fitting should include some regularization. The natural way to introduce regularization in this context is to add artificial diffusion such that we are really solving for

$$-\nabla \cdot D \nabla f + f = \hat{f} \quad (59.1)$$

which is a linear diffusion-reaction equation. The weak formulation for this is

$$\int D \nabla f \cdot \nabla \psi \, d\Omega - \int D \frac{\partial f}{\partial n} \, d\Gamma \int f \, d\Omega = \int \hat{f} \, d\Omega \quad (59.2)$$

Now the data could be given already in the integral form i.e. as

$$\hat{F} = \int \hat{f} \, d\Omega. \quad (59.3)$$

Also the user may know the weight w that was accumulated when the data was generated.

The data may not always be accurate or even given. Therefore using the given data in all fields might not be a good idea. We may introduce a masking field, say $p \in [0, 1]$, that picks the values that are considered as accurate. On the other hand diffusion, might not be needed for those points.

Adding these generalizations we obtain the following form

$$\int (1-p) D \nabla f \cdot \nabla \psi \, d\Omega - \int D(1-p) \frac{\partial f}{\partial n} \psi \, d\Gamma + \int p w f \psi \, d\Omega = \int p \hat{f} \psi \, d\Omega + p \hat{F}. \quad (59.4)$$

This formulation enables the use of various techniques for data fitting.

59.3 Keywords

Below are the dedicated keywords for the solver. In addition generic keywords such as many of those related to Linear System solution methods may be used.

Solver solver id

Equation String [DataToFieldSolver]

The name of the equation.

Procedure File "DataToFieldSolver" "DataToFieldSolver"

The name of the procedure.

Variable String [FieldName]

The name of the resulting field.

Target Variable String

Name of the scalar field to be used as input data. This is either \hat{f} or \hat{F}

Diffusion Coefficient Real

The value of the diffusion constant used in the regularization.

Normalize by Nodal Weight Logical

Normalize the particle properties with nodal weight i.e. the standard mass matrix. This would be a good choice if the right-hand-side is a result of an integral i.e. \hat{F} .

Normalize by Given Weight Logical

Normalize the particle properties i.e. divide the trace with the corresponding weight. Basically this corresponds to the integral over w . Here an integral data set is assumed such that it is consistent with the given weight.

Weight Variable String

If there is a weight associated to the data the name of it should be given with this function.

Set Constant Weight Sum Logical

If the weights are provided by the user and this flag is set True then the given weights are normalized so that their sum is always the same as the sum of nodal weights. The idea is that the solution will be independent on the given level of the weight, and only depend on the relative sizes.

Mask Variable T

The data points may selectively chosen to be considered. If a mask variable is given it may have both lower and upper boundaries. Only if the mask variable is between these bounds will the data be considered. Effectively $p = 1$ in the interval, and $p = 0$ outside the interval.

Max Mask Value Real

Maximum value for the mask interval. Default is +HUGE.

Min Mask Value Real

Minimum value for the mask interval. Default is -HUGE, except if the maximum interval is not given either. Then the default is 0.

Mask Diffusion Logical

If the mask is true, should diffusion be neglected. Default is False.

Boundary Condition bc id

FieldName Real

The user may set Dirichlet conditions for the field to be fitted if they are known.

FieldName Continue Logical

Enforce boundary conditions with constant slope. The default boundary condition is otherwise the natural boundary condition with zero gradient.

Model 60

Projection to plane

Module name: ProjectToPlane

Module subroutines: ProjectToPlane, ParallelProjectToPlane

Module authors: Juha Ruokolainen and Peter Råback

Document authors: Peter Råback

Document edited: 18.6.2010

60.1 Introduction

Sometimes the solution of a complex problem calls for a dimensional reduction of some field variables. A possible scenario for using the solver is in extracting some useful information from DNS or LES type of flow simulations. This module offers the subroutines needed in such a cases.

There are currently a serial and a parallel version of the subroutine which are both located in this module. The reason for the fork is that the parallel version uses techniques that are not optimal in the serial problem. Therefore the user should herself choose the correct version. Optimally these two approaches should of course be fused.

60.2 Theory

In principle the dimensional reduction is performed taking on average of a 2D (or axisymmetric) nodal point \vec{r}_{2D} when it travels through the 3D mesh.

$$f_{2D}(\vec{r}_{2D}) = \frac{1}{S} \int f(\vec{r}_{3D}) dS. \quad (60.1)$$

In practice this is implemented with the following steps

1. Create a list of faces for the 3D mesh
2. Loop though each nodal point in the 2D mesh
 - (a) Loop through each face in the 3D mesh
 - i. Check if there is an intersection between the integration line and face
 - ii. If intersection found memorize the point of intersection
 - (b) Order the intersection points in the integration direction
 - (c) Take an weighted average over the ordered list, (f_i, r_i)

The algorithm is accurate for linear elements. For higher order elements it is suboptimal in accuracy. Also in axisymmetric mapping the elements should be small enough so that the curvature of the line segment is not significant. Near the origin there may be few hits and then the averaging is done by just taking a small number of values around the center axis.

60.3 Keywords

Solver `solver id`

Equation `ProjectToPlane`

The arbitrary name of the equation.

Procedure `File "ProjectToPlane" "ProjectToPlane"`
or

Procedure `File "ProjectToPlane" "ParallelProjectToPlane"`

Convert From Equation Name `String`

The solver needs a 3D mesh which is associated determined by the association to the solver given by this keyword.

Convert From Variable `String`

The variable to be converted.

Volume Permutation `Integer`

The algorithm is build so that integration direction is the second coordinate (y). This is typically valid for axisymmetric cases, for example. If the integration should be performed with respect to some other direction the volume coordinates may be permuted by this keyword.

Plane Permutation `Integer`

Permutation of the plane coordinates.

Rotate Plane `Logical`

Should rotation be performed.

Max Relative Radius `Real`

For the axisymmetric projection the outer radius may be difficult since the 3D mesh typically may have faces that do not quite extend to the surface. This is a result of finite sized linear elements. To ease this problem the user may give the maximum relative radius that is used when trying to find the point of intersection.

Minimum Hits At Radius `Integer`

The number of hits needed for a accepted integration. The default is one.

Integration Points At Radius `Integer`

If no minimum number of hits is achieved then a few points around the axis is used to determine the value. The default is two.

Model 61

Structured projection to plane

Module name: StructureProjectToPlane
Module subroutines: StructureProjectToPlane
Module authors: Peter Råback
Module status: Beta
Document authors: Peter Råback
Document created: 4.4.2011
Document edited: 20.5.2016

61.1 Introduction

For structured meshes some operations may be done much more effectively than for generic meshes. One such operation is mapping some values within the mesh to the reduced dimensional surface of the mesh. In practice this could mean, for example, mapping the values at the bottom of mesh to the top of mesh to determine the difference in value. Or to map some isosurface values to the top of the mesh.

61.2 Theory

The algorithm used for the mapping has two sweeps. Assume that we would like to perform mapping in direction \vec{e}_z . The at the first sweep over all elements we would deduce pairwise information over nodes on which nodes are in `up` and `down` directions from each other. Then using this directional information recursively one can easily deduce which nodes are the `top` and `bottom` representatives of any node in the mesh. With this information mapping information to top or bottom nodes becomes extremely cheap.

61.3 Keywords

`Solver` `solver id`

`Equation` `String` [`StructuredProjectToPlane`]
The name of the equation.

`Procedure` `File` "`StructuredProjectToPlane`" "`StructuredProjectToPlane`"
The name of the procedure.

`Velocity Variable Name` `String`
Name of the variable used to define the streamlines.

`Active Coordinate` `Integer`
The direction in which the stuctured mapping is performed i.e. 1, 2 or 3.

`Project To Bottom Logical`

Instead of projecting to the top of the active direction, project to the bottom.

`Projection Mask Variable String`

By default the projection is performed assuming that the whole mesh is active. This keyword may be used to choose the variable that chooses the active set of nodes user in the projections.

`Dot Product Tolerance Real`

When determining the structure of the mesh in the active direction this tolerance is used to decide that an element edge is aligned with the direction of the action.

`Variable i String`

When applying the different projections, the variable to apply the projection to. The resultant field will obtain as the name the name of the operator followed by the initial variable name.

`Operator i String`

The operator to apply to the variable or just the geometry. The choices are

- `sum`: sum over all nodes on the structured line
- `int`: integral over the structured line using trapetsoidal rule
- `min`: minimum value over the line
- `max`: maximum value over the line
- `bottom`: field value at the bottom layer
- `top`: field value at the top layer
- `middle`: field value at the middle layer
- `thickness`: thickness of the object i.e. the length of the line
- `depth`: depth i.e. distance from the top surface
- `height`: height i.e. distance from the bottom surface
- `distance`: minimum distance to either top or bottom surface
- `index`: number of the layer from the bottom
- `layer below top`: field value at layer below the top
- `layer above bottom`: field value at layer above bottom
- `isosurface`: field value at the given value of the isosurface variable

`Layer Index i Integer`

The number of structured layers from the top or bottom layer when appropriate. The index `i` refers to the corresponding variable and operator.

`Isosurface Variable i String`

The variable used to determine the isosurface position.

`Isosurface Value i Real`

The value for the isosurface position.

Model 62

Statistics of finite element mesh

Module name: ElementStats

Module subroutines: ElementStats

Module authors: Peter Råback

Module status: alpha

Document authors: Peter Råback

Document created: 17.5.2017

Document edited: 17.5.2017

62.1 Introduction

This module is used to calculate provide information on the mesh. The quality of the mesh will have a great effect on the solution. This solver may help the user to determine whether the mesh is suitable for the need.

Currently three different operations are performed:

- element size
This is the metric determinant given by the ElementInfo.
- element skew
This is the skewness (in degrees) of the mesh, if applicable. Only quadrilaterals, pyramids, wedges and hexahedrons may be skewed.
- element ratio
This refers to the ratio between maximum and minimum edge lengths of an element.

The operations are performed separately for bulk and boundary elements.

It is available currently only in serial.

62.2 Keywords

Solver `solver id`

Equation `String ElementStats`

Procedure `File "ElementStats" "ElementStats"`

Create Histogram `Logical`

When analyzing the mesh the results may be shown also as a histogram with an even distribution between min and max values. This flag activates the classification of the statistics into a histogram.

Histogram Intervals `Integer`

The number of intervals in the histogram when classifying the element properties.

Part X

Experimental or Obsolete Solvers

Model 63

Richards equation for variably saturated porous flow

Module name: RichardsSolver

Module subroutines: RichardsSolver, RichardsFlux

Module authors: Peter Råback

Module status: Alpha

Document authors: Serge-Étienne Parent and Peter Råback

Document created: 22.12.2010

Document edited: 22.12.2010

63.1 Introduction

Richards equation is a non-linear partial differential equation that represents the movement of fluids through porous media.

The current implementation of the Richards equation uses normal Lagrange elements and therefore the conservation of flux cannot be guaranteed. Dense meshes are required if the variations in the permeability are high.

This version should not yet be considered a production version. However, it provides a suitable starting case for more serious testing and further development.

63.2 Theory

The transient, incompressible, variably saturated, isotropic flow of water in non-swelling porous media is expressed by the combination of Darcy's law and the continuity equation, i.e. Richards equation. The modern form of Darcy's law can be written as

$$\vec{q} = -k_w \nabla H \quad (63.1)$$

where \vec{q} is the unit flux, or Darcy velocity (L/T), k_w is the fluid hydraulic conductivity of water (L/T), and H is the total head (L). Since the velocity component of total head can be treated as negligible in porous media, and air pressure can be considered as constant, total head can be expressed as $H = p + z$ where p is pressure (F/L²) (note that $p = -\psi = u_a - u_w$), ψ is matric suction (F/L²), u_w is the water pressure in pores (F/L²), u_a is the air pressure in pores (F/L²), z is elevation from a datum (depth coordinate of the geometry).

The continuity equation is expressed as

$$\frac{\partial \theta}{\partial t} = -\nabla \cdot \vec{q} + S_w, \quad (63.2)$$

where θ is the volumetric water content (L/L), t is time (T), S_w is a source/sink term (L/T). Richards equation may now be written as,

$$\frac{\partial \theta}{\partial t} = -\nabla \cdot (k_w \nabla H) + S_w, \quad (63.3)$$

since $\theta = f(\psi)$ and $k_w = f(\psi)$ the latter equation can be expressed using a pressure form,

$$\frac{\partial \theta}{\partial \psi} \frac{\partial \psi}{\partial t} = -\nabla \cdot (k_w \nabla H) + S_w, \quad (63.4)$$

Or, developing total head,

$$\frac{\partial \theta}{\partial \psi} \frac{\partial \psi}{\partial t} = -\nabla \cdot (k_w \nabla (-\psi + z)) + S_w, \quad (63.5)$$

The volumetric water content and the hydraulic conductivity are non-linear functions related to pressure head. Both are commonly expressed by van Genuchten (1980)'s equations. Volumetric water content function yields

$$\theta(\psi) = \begin{cases} \theta_s + \frac{\theta_s - \theta_r}{(1 + \alpha_{vG} \psi)^{m_{vG}}}, & \text{if } \psi > 0 \\ \theta_r, & \text{if } \psi < 0. \end{cases} \quad (63.6)$$

And the hydraulic conductivity function is

$$k_w(\psi) = \begin{cases} k_{w,sat} \frac{(1 - (\alpha_{vG} \psi)^{n_{vG}})^{m_{vG}} (1 + (\alpha_{vG} \psi)^{n_{vG}})^{m_{vG}}}{(1 + \alpha_{vG} \psi)^{m_{vG}/2}}, & \text{if } \psi > 0 \\ k_{w,sat}, & \text{if } \psi < 0. \end{cases} \quad (63.7)$$

where θ is the volumetric water content (L/L), θ_r is the residual volumetric water content (L/L), θ_s is the saturated volumetric water content, equal to the porosity (L/L), α_{vG} , n_{vG} , m_{vG} are fitting parameters without any units.

63.3 Implementation issues

The current implementation is carried out for the total head, H . This results to a weak form where the fluxes occur naturally. The total head is intuitive since it gives directly the ground water level. Since the time derivative of the elevation is zero, we may use the following equation to solve the total head,

$$\theta_\psi \frac{\partial H}{\partial t} + \nabla \cdot (k_w \nabla H) = S_w. \quad (63.8)$$

From the total head the matric suction will be automatically computed, $\psi = z - H$. This makes it possible to have material laws that depend on it.

For transient problems the first term requires special attention. In the current version the sensitivity of θ to ψ is computed from

$$\theta_\psi = \begin{cases} \frac{\theta(\psi(t_i)) - \theta(\psi(t_{i-1}))}{\psi(t_i) - \psi(t_{i-1})} & \text{if } |\psi(t_i) - \psi(t_{i-1})| > \epsilon \\ \frac{\theta(\psi(t_i)) - \theta(\psi(t_i) - \epsilon)}{\epsilon} & \text{otherwise.} \end{cases} \quad (63.9)$$

This way the effective sensitivity is smeared over the whole timestep, $dt = t_i - t_{i-1}$.

The values of the material parameters in the Richards equation vary a great deal depending on the saturation level and type of medium. Therefore it is important to evaluate the water content and hydraulic conductivity at the Gaussian integration points using the relevant formulas, rather than computing them at nodal points and thereafter evaluating the values at the Gaussian integration points using a weighted sum over the nodal values.

63.4 Keywords

The module includes two different solvers. `RichardsSolver` solves the primary differential equation while `RichardsFlux` solves the resulting flux from the computed solution. The second solver only makes sense when the pressure field has already been computed with the first one. The second solver uses the same material parameters as the first one.

Keywords for RichardsSolver

Solver `solver id`

Equation `String RichardsSolver`

A describing name for the solver. This can be changes as long as it is used consistently.

Procedure `File "RichardsSolver" "RichardsSolver"`

Name of the solver subroutine.

Variable `String TotalHead`

The name of the variable may be freely chosen as far as it is used consistently also elsewhere.

Variable DOFs `Integer 1`

Degrees of freedom for the pressure. This should be 1 which is also the default value.

Saturated Initial Guess `Logical`

Use saturated material parameters when computing the first equation for the total head.

Active Coordinate `Integer`

The coordinate corresponding to the depth z in the Richards equation. By default the last coordinate is the active one.

Calculate Matrix Suction `Logical`

Whether to compute the matric suction from the total head.

Bubbles `Logical`

Use stabilization by residual free bubbles.

Nonlinear System Convergence Tolerance `Real`

The Richards equation is always nonlinear and hence keywords related to the nonlinear system control are needed. The iteration of the nonlinear system is continued till the relative change in the norm falls under the value given by this keyword.

Nonlinear System Max Iterations `Integer`

This parameter gives the maximum number of nonlinear iterations required in the solution. This may be set higher than the typical number of iterations required as the iteration procedure should rather be controlled by the convergence tolerance.

Nonlinear System Relaxation Factor `Real`

Keyword related to the relaxation of the nonlinear system.

Material `mat id`

Porosity Model `String`

Currently the choices are `van Genuchten` and `Default`. The latter does not estimate the functional forms on gaussian points and hence may have enferior accuracy. Also, currently the computation of water content derivative is not supported for it limiting its usability to steady state problems.

Saturated Hydraulic Conductivity `Real`

Saturated Water Content `Real`

Residual Water Content `Real`

van Genuchten Alpha `Real`

van Genuchten N `Real`

van Genuchten M `Real`

The parameters above are the material parameters of the van Genuchten material law that are used to compute the hudraulic conductivity and water content.

Hydraulic Conductivity `Real`

Water Content `Real`

In case the porosity model is `constant` then the hydraulic conductivity and water content are given with this keyword.

Body Force `bf id`

Richards Source `Real`

The source term, S_w , of the equation.

Boundary Condition `bc id`

Richards Flux `Real`

The given flux at the boundary.

Keywords for RichardsPostprocess

This solver uses largely the same keywords that are already defined above. Only the Solver section has its own keyword settings. This solvers should be active in the same bodies than the RichardsSolver.

Solver `solver id`

Equation `String RichardsPostprocess`

A describing name for the solver. This can be changes as long as it is used consistently.

Procedure `File "RichardsSolver" "RichardsPostprocess"`

Name of the solver subroutine.

Target Variable `String`

The name of the total head field solved by the Richards equation. The default name is Total head.

Model 64

BEM Solver for Poisson Equation

Module name: PoissonBEM

Module subroutines: PoissonBEMSolver

Module authors: Juha Ruokolainen

Document authors: Juha Ruokolainen

Document edited: May 27th 2003

64.1 Introduction

This module solves the Laplace equation by boundary element method (BEM), where the differential equation is transformed to integral equation along the boundaries. On the boundaries either potential or normal flux may be defined. A source term may be included (Poisson equation), but the source term remains a volume integral.

64.2 Theory

The Poisson equation is mathematically described as

$$-\Delta\Phi - f = 0, \text{ in } \Omega, \quad (64.1)$$

where f is the given source.

In BEM we transform this equation to integral equation over boundaries. We start by multiplying the equation by a weight function and integrating over the volume, and integrating by parts

$$-\int_{\Omega} \Delta\Phi w \, d\Omega = \int_{\Omega} \nabla\Phi \cdot \nabla w \, d\Omega - \int_{\Gamma} \frac{\partial\Phi}{\partial n} w \, d\Gamma. \quad (64.2)$$

Similarly we may write an equation reversing the roles of Φ and w

$$-\int_{\Omega} \Delta w \Phi \, d\Omega = \int_{\Omega} \nabla w \cdot \nabla\Phi \, d\Omega - \int_{\Gamma} \frac{\partial w}{\partial n} \Phi \, d\Gamma. \quad (64.3)$$

Subtracting the two equations we have

$$-\int_{\Omega} \Delta\Phi w \, d\Omega = -\int_{\Omega} \Delta w \Phi \, d\Omega - \int_{\Gamma} \frac{\partial\Phi}{\partial n} w \, d\Gamma + \int_{\Gamma} \frac{\partial w}{\partial n} \Phi \, d\Gamma \quad (64.4)$$

Next we choose the weight w as follows:

$$-\Delta w = \delta_r(r'), \quad (64.5)$$

so that

$$-\int_{\Omega} \Delta w \Phi \, d\Omega = \Phi(r), \quad (64.6)$$

The weight w chosen this way is the Green's function for the Laplace operator, i.e.

$$w(r, r') = \frac{\log(r - r')}{2\pi} \text{ in 2d}, w(r, r') = \frac{1}{4\pi(r - r')} \text{ in 3d}. \quad (64.7)$$

Finally we add the source term, and we have the equation

$$\Phi(r) - \int_{\Gamma} \frac{\partial \Phi}{\partial n} w \, d\Gamma + \int_{\Gamma} \frac{\partial w}{\partial n} \Phi \, d\Gamma - \int_{\Omega} f w \, d\Omega = 0. \quad (64.8)$$

Only the source term is now integrated over the volume. This equation may now be discretized by standard methods.

64.2.1 Boundary Conditions

Boundary conditions may be set for either potential

$$\Phi = \Phi_{\Gamma} \text{ on } \Gamma, \quad (64.9)$$

or normal flux

$$-\frac{\partial \Phi}{\partial n} = g \text{ on } \Gamma. \quad (64.10)$$

64.3 Keywords

`Solver solver id`

Note that all the keywords related to linear solver (starting with `Linear System`) may be used in this solver as well. They are defined elsewhere. Note also that the BEM discretization results to a full linear system in contrast to FEM discretizations and the ILU preconditioning settings are not available.

`Equation String [PoissonBEM]`

The name of the equation.

`Procedure File ["PoissonBEM" "PoissonBEMSolver"]`

This keyword is used to give the Elmer solver the place where to search for the equation solver.

`Variable String [Potential]`

Give a name to the field variable.

`Variable DOFs Integer [1]`

This keyword must be present, and *must* be set to the value 1.

`Exported Variable 1 String Flux`

If this keyword is given, the output will include the normal flux at boundaries, the name must be exactly as given.

`Exported Variable 1 DOFs Integer [1]`

This keyword must be present if Flux values are to be computed, and *must* be set to the value 1.

`Equation eq id`

The equation section is used to define a set of equations for a body or set of bodies:

`PoissonBEM Logical`

if set to True, solve the Poisson equation, the name of this parameter must match the `Equation` setting in the `Solver` section.

If the mesh has any volume elements with a body id that corresponds to a body where the Poisson equation is activated, the value of the potential is computed for these elements as a postprocessing step. Note that the computation of potential is not a trivial task, so large number of volume elements may result to long execution time.

Boundary Condition `bc id`

The boundary condition section holds the parameter values for various boundary condition types. Dirichlet boundary conditions may be set for all the primary field variables. The one related to Poisson (BEM) equation are

Body Id `Integer`

Give body identification number for this boundary, used to reference body definitions in .sif file. This parameter must be set so that the ElmerSolver knows at which boundaries to solve the corresponding equation.

Potential `Real`

Known potential value at boundary.

Flux `Real`

Known normal flux at boundary.

Normal Target Body `Integer`

The direction of boundary normals are important for the success of the computation. They should point consistently outward from the boundaries. This is accomplished either if the mesh generator automatically orients the boundary elements consistently, or including in the mesh the parent (volume) elements of the boundaries and using this keyword. The value -1 of this parameter points to the side where there are no volume elements. If the parameter gets the value of the body id of the volume elements, the normal will point to that direction.

Body Force `bf id`

The source term for the Poisson equation may be given here. The volume integral is computed on a body with a volume mesh and the PoissonBEM equation set to true.

Source `Real`

The source term for the Poisson equation.

Model 65

BEM Solver for Helmholtz Equation

Module name: HelmholtzBEM

Module subroutines: HelmholtzBEMSolver

Module authors: Juha Ruokolainen

Document authors: Juha Ruokolainen

Document edited: May 27th 2003

65.1 Introduction

This module solves the Helmholtz equation by boundary element method (BEM), where the differential equation is transformed to integral equation along the boundaries. On the boundaries either pressure or normal flux may be defined.

65.2 Theory

The Helmholtz equation is mathematically described as

$$(k^2 + \Delta)\Phi = 0, \text{ in } \Omega. \quad (65.1)$$

In BEM we transform this equation to integral equation over boundaries. We start by multiplying the equation by a weight function and integrating over the volume, and integrating by parts

$$\int_{\Omega} (k^2 + \Delta)\Phi w \, d\Omega = \int_{\Omega} k^2 w \Phi \, d\Omega - \int_{\Omega} \nabla \Phi \cdot \nabla w \, d\Omega + \int_{\Gamma} \frac{\partial \Phi}{\partial n} w \, d\Gamma. \quad (65.2)$$

Similarly we may write an equation reversing the roles of Φ and w

$$\int_{\Omega} (k^2 + \Delta)w \Phi \, d\Omega = \int_{\Omega} k^2 w \Phi \, d\Omega - \int_{\Omega} \nabla w \cdot \nabla \Phi \, d\Omega + \int_{\Gamma} \frac{\partial w}{\partial n} \Phi \, d\Gamma. \quad (65.3)$$

Subtracting the two equations we have

$$\int_{\Omega} (k^2 + \Delta)\Phi w \, d\Omega = \int_{\Omega} (k^2 + \Delta)w \Phi \, d\Omega - \int_{\Gamma} \frac{\partial \Phi}{\partial n} w \, d\Gamma + \int_{\Gamma} \frac{\partial w}{\partial n} \Phi \, d\Gamma \quad (65.4)$$

Next we choose the weight w as follows:

$$(k^2 + \Delta)w = \delta_r(r'), \quad (65.5)$$

so that

$$\int_{\Omega} (k^2 + \Delta)w \Phi \, d\Omega = \Phi(r), \quad (65.6)$$

The weight w chosen this way is the Green's function for the Helmholtz operator, i.e.

$$w(r, r') = \frac{1}{i4} H_0(k(r - r')) \text{ in 2d, } w(r, r') = \frac{1}{4\pi} \exp^{-ik(r-r')} \text{ in 3d,} \quad (65.7)$$

where H_0 is the Hankel function.

Finally we have the equation

$$\Phi(r) - \int_{\Gamma} \frac{\partial \Phi}{\partial n} w \, d\Gamma + \int_{\Gamma} \frac{\partial w}{\partial n} \Phi \, d\Gamma = 0. \quad (65.8)$$

65.2.1 Boundary Conditions

Boundary conditions may be set for either pressure

$$\Phi = \Phi_{\Gamma} \text{ on } \Gamma, \quad (65.9)$$

or normal flux

$$-\frac{\partial \Phi}{\partial n} = g \text{ on } \Gamma. \quad (65.10)$$

65.3 Keywords

Simulation

Angular Frequency Real

Give the value of the angular frequency for the simulation.

Solver solver id

Note that all the keywords related to linear solver (starting with Linear System) may be used in this solver as well. They are defined elsewhere. Note also that the BEM discretization results to a full linear system in contrast to FEM discretizations and the ILU preconditioning settings are not available.

Equation String [HelmholtzBEM]

The name of the equation.

Procedure File ["HelmholtzBEM" "HelmholtzBEMSolver"]

This keyword is used to give the Elmer solver the place where to search for the equation solver.

Variable String [Pressure]

Give a name to the field variable.

Variable DOFs Integer [2]

This keyword must be present, and *must* be set to the value 2.

Exported Variable 1 String Flux

If this keyword is given, the output will include the normal flux at boundaries, the name must be exactly as given.

Exported Variable 1 DOFs Integer [2]

This keyword must be present if Flux values are to be computed, and *must* be set to the value 2.

Equation eq id

The equation section is used to define a set of equations for a body or set of bodies:

HelmholtzBEM Logical

if set to True, solve the Helmholtz equation, the name of this parameter must match the Equation setting in the Solver section.

If the mesh has any volume elements with a body id that corresponds to a body where the Helmholtz equation is activated, the value of the pressure is computed for these elements as a postprocessing step. Note that the computation of potential is not a trivial task, so large number of volume elements may result to long execution time.

Boundary Condition `bc id`

The boundary condition section holds the parameter values for various boundary condition types. Dirichlet boundary conditions may be set for all the primary field variables. The one related to Helmholtz (BEM) equation are

Body Id `Integer`

Give body identification number for this boundary, used to reference body definitions in .sif file. This parameter must be set so that the ElmerSolver knows at which boundaries to solve the corresponding equation.

Pressure 1 `Real`

Known real part of pressure at boundary.

Pressure 2 `Real`

Known imaginary part of pressure at boundary.

Flux 1 `Real`

Known real part of normal flux at boundary.

Flux 2 `Real`

Known real part of normal flux at boundary.

Normal Target Body `Integer`

The direction of boundary normals are important for the success of the computation. They should point consistently outward from the boundaries. This is accomplished either if the mesh generator automatically orients the boundary elements consistently, or including in the mesh the parent (volume) elements of the boundaries and using this keyword. The value -1 of this parameter points to the side where there are no volume elements. If the parameter gets the value of the body id of the volume elements, the normal will point to that direction.

Model 66

Outlet Boundary Condition for Arterial Flow Simulations

Module name: ArteryOutlet

Module subroutines: OutletCompute, OutletInit, OutletPres, OutletdX, OutletdY

Module authors: Esko Järvinen, Mikko Lyly, Peter Råback

Document authors: Esko Järvinen

Document created: April 28th 2006

Document edited: April 28th 2006

66.1 Introduction

Arterial elasticity is a fundamental determinant of blood flow dynamics in arteries, such as the aorta and its daughter vessel, that face the largest displacements and which takes care of the cushioning of the stroke volume. Simulation of such a phenomenon requires simultaneous solving of the equations governing both the fluid flow and wall elasticity. To be able to perform accurate fluid-structure interaction (FSI) simulations, only a segment of the circulatory system can be studied at a time. For these artificially truncated segments, which are naturally unbounded domains and in interaction with the rest of the circulation domain, one should construct in the numerical models boundary conditions which do not exhibit any unphysical behaviour, which operates transparently, and are also capable to transport a sufficient amount of information over the boundary.

A natural boundary condition at the outlet of a numerical FSI flow model of an artery is not a proper choice because it does not exhibit enough correct physiological behavior of the flow, and from the point of view of numerical approximation, it causes non-physiological reflections of the wave at the boundary. If measured data of both the pressure (or velocity) and the wall displacement at the outlet boundary are not available, the only way to get the outlet boundary of a higher order, 2D or 3D model sufficiently specified is to combine the model with some lower order model, such as a 1D or lumped model.

In order to get the outlet of the arterial FSI model to behave transparently in such cases when only forward travelling waves are considered, a simple characteristic equation of the of the one dimensional FSI model can be combined with the higher order model.

66.2 Theory

The conservation equations for a flow in an elastic artery in one dimension may be expressed as

$$\begin{cases} \frac{\partial A}{\partial t} + \frac{\partial Q}{\partial x} = 0 \\ \frac{\partial Q}{\partial t} + \frac{\partial}{\partial x} \left(\frac{Q^2}{A} \right) + \frac{A}{\rho} \frac{\partial p}{\partial x} = 0, \end{cases} \quad (66.1)$$

where Q is the volume flow, A the cross section area of the artery, p is the pressure and x is the axial coordinate [3]. In order to get the system (66.1) close, an equation relating the area A to the pressure $p = p(A)$ is derived applying the theory of thin shell structures. Assuming a cylindrical shell, and neglecting the rotation on the shell cut plane, and the movements of the structure in the axial and circumferential directions, as well as applying the Kirchhoff-Love assumption, the energy balance equations is reduced to

$$\frac{E h^3}{12(1-\nu^2)} d_R^{(4)} + \frac{E h}{(1-\nu^2)} \frac{1}{R_m^2} d_R = p, \quad (66.2)$$

where R_m is the radius to the midplane of the wall, E , ν and d_R are the Young's modulus, the Poisson ratio and the radial displacement of wall, respectively. Assuming that the first term on the left side in the equation (66.2) is much smaller than the second term, we can give the pressure-area relation in the form

$$p = p_{ext} + \beta(\sqrt{A} - \sqrt{A_0}), \quad \beta = \frac{\sqrt{\pi} h E}{(1-\nu^2) A_0}. \quad (66.3)$$

The pressure is scaled to be equal to external pressure p_{ext} with corresponding reference artery cross sections area A_0 .

The equations (66.1) and (66.3) form a closed system for the simulations of flow in an elastic tubes. The equations may be written in conservative form which is strictly hyperbolic with two distinct real eigenvalues $\lambda_{1,2} = \bar{u} \pm c$, where $\bar{u} = Q/A$ is the average axial velocity, $c = \sqrt{(A/\rho_f)(\partial p/\partial A)} = \sqrt{\beta\sqrt{A}/(2\rho_f)}$ is the speed of sound, and ρ_f is the density of blood. The system can be further decomposed into a set of the equations for the characteristic variables W_i , which are the components of the vector $W = T^{-1}U$ ($\frac{\partial W}{\partial t} = T^{-1}U$), $U = [A, Q]^T$ [2]. These equations are

$$\frac{\partial W_i}{\partial t} + \lambda_i \frac{\partial W_i}{\partial x} = 0, \quad (66.4)$$

and the characteristic variables are

$$W_{1,2} = \frac{Q}{A} \pm 2\sqrt{\frac{2}{\rho_f} + \beta\sqrt{A}}.$$

When considering a pulse propagation in a straight, infinitely long homogeneous conduit, without any bifurcations or other objects which might cause reflections of the pulse, i.e. any backward travelling waves does not exist, the computations can be done using only the first of equations in (66.4), i.e.

$$\frac{\partial W_1(U)}{\partial t} + \lambda_1(U) \frac{\partial W_1(U)}{\partial x} = 0.$$

This equation is solved in this Elmer outlet boundary condition for arterial flow simulations solver. The connection of the one dimensional model to the test models at their outlets is done applying the following coupling [1]

$$\begin{cases} dR^- = dR^+ \\ \sigma^- = p^+ \\ W_1 = g_1(A^-, Q^-, p^-), \end{cases}$$

where dR and σ are radial displacement of the artery wall and fluid traction, respectively. The superscript '-' denotes the values in the higher order models, and superscript '+' to the values in the 1D model.

66.3 Keywords

Keywords of FlowSolve

Initial Condition `ic id`

For making the initial guess for the characteristic variable W_1

```
Wnodal Variable Coordinate
      Real Procedure "ArteryOutlet" "OutletInit"
```

Material mat id
Material properties for the one dimensional section.

```
Density Real
      Density of blood
Artery Wall Youngs Modulus Real
      Young's modulus of the artery
Artery Radius Real
      Radius of the artery to the midplane of the artery wall
Artery Wall Thickness Real
      Wall thickness of the artery
Artery Poisson Ratio Real
      Poisson ration of the artery
```

Solver solver id
Keywords for the one dimensional solver. Note that all the keywords related to linear solver (starting with Linear System) may be used in this solver as well. They are defined elsewhere.

```
Equation String [Artery Outlet Solver]
```

```
Variable [Wnodal]
      The variable which is solved
Variable DOFs [1]
```

```
Procedure File "ArteryOutlet" "OutletCompute"
      The name of the file and the subroutine
```

Equation eq id
The equation section is used to define a set of equations for a body or set of bodies

```
Artery Outlet Solver Logical [True]
      If set True, the solver is used. The name of the solver must match with the name in the Solver
      section
```

Boundary Condition boundary id
The pressure of the given coordinate direction i at the artery outlet of the higher order model is set to correspond the value given by the 1D model.

```
Pressure i Variable Time
      Real Procedure "ArteryOutlet" "OutletPres"
```

The diameter of the artery in the appropriate direction at the outlet of the higher order model is set to correspond the value given by the outlet boundary condition solver. The subroutines OutletdX and OutletdY are located in the module ArteryOutlet

```
Displacement i Variable Time
      Real Procedure "./ArteryOutlet" "OutletdX"
```

This is the inlet boundary of the one dimensional section which is coupled with both, the fluid and the solid outlet boundary of the higher order model

```
Fluid Coupling With Boundary Integer
Structure Coupling With Boundary Integer
```

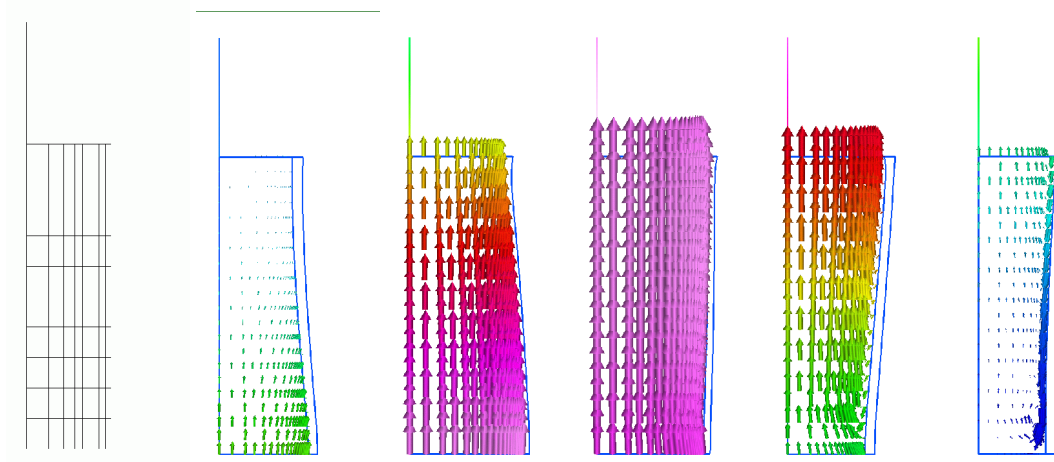


Figure 66.1: An example of the model results: pressure pulse propagation in a 2D axisymmetric model combined with an 1D model.

Bibliography

- [1] L. Formaggia, J.F. Gerbeau, F. Nobile, and A. Quarteroni. Numerical treatment of defective boundary conditions for the navier-stokes equations. *SIAM J. Numer. Anal.*, 40:376–401, 2002.
- [2] E. Godlewski and P.-A. Raviart. *Numerical Approximation of Hyperbolic Systems of Conservation Laws*. Springer, 1996.
- [3] T. J. R. Hughes and J. Lubliner. On the one-dimensional theory of blood flow in the large vessels. *Mathematical Biosciences*, 19:161–170, 1973.

Model 67

Magnetoquasistatic approximation for axial symmetry

Module name: StatMagSolve

Module subroutines: StatMagSolver

Module authors: Juha Ruokolainen, Ville Savolainen, Jussi Heikonen, Peter Råback, Antti Pursula

Document authors: Ville Savolainen, Peter Råback, Antti Pursula, Mika Malinen

Document created: 29.6.2006

Document edited: 30.5.2013

67.1 Introduction

Note: This solver is obsolete!

Maxwell's equations may generally be expressed by employing a scalar potential and a vector potential. The magnetic flux density is then the curl of the vector potential. In some cases the effect of the scalar potential vanishes and the system is fully described by the vector potential. These cases include magnetostatics problems where time-independent magnetic fields may be created by electromagnets with given current distributions or permanent ferromagnets. The solver considered here allows the first option, with non-homogeneous and non-linear magnetic materials.

The scalar potential may also be ignored in two-dimensional magnetoquasistatic cases when the current density acts in a direction orthogonal to the plane considered. Then eddy current effects relating to a sinusoidal evolution of the current density may also be considered at low frequencies. If there are no conductors in the system, this approximation reduces to the equations of magnetostatics.

This solver was historically developed for the axially symmetric cases and it should only be used in those. For handling problems in orthogonal Cartesian coordinates, see the modules `MagnetoDynamics` and `MagnetoDynamics2D` for 3-D and 2-D versions, respectively.

67.2 Theory

When there are no hard ferromagnets, a magnetostatics problem may be expressed using the magnetic vector potential \vec{A} that gives the magnetic flux density as $\vec{B} = \nabla \times \vec{A}$. It is obtained directly using the Ampère's law, so that

$$\nabla \times \left(\frac{1}{\mu} \nabla \times \vec{A} \right) = \vec{j}. \quad (67.1)$$

Here μ is the magnetic permeability of the material. The equation may be non-linear through the magnetic permeability curve of a ferromagnetic material. The solver discussed is intended for handling the axially symmetric version of (67.1).

The axially symmetric version of (67.1) may also be employed to handle magnetoquasistatic problems where the effect of the displacement current is ignored. If there are conductors in the system, the current density is then written as $\vec{j} = \sigma \vec{E} + \vec{j}_0$ where σ is the electric conductivity and the electric field \vec{E} is given by

$$\vec{E} = -\frac{\partial \vec{A}}{\partial t}.$$

In the time-harmonic case the source current density \vec{j}_0 is considered to be $\vec{j}_0(x, t) = \vec{j}_0(x) e^{i\omega t}$, where $\omega = 2\pi f$ is the angular frequency. Using a trial $\vec{A}(x, t) = \vec{A}_0(x) e^{i\omega t}$, we then obtain an equation for the amplitude:

$$\nabla \times \left(\frac{1}{\mu} \nabla \times \vec{A}_0 \right) + i\omega \sigma \vec{A}_0 = \vec{j}_0.$$

In the axially symmetric case, the magnetic flux density \vec{B} has only r - and z -components, while the current density \vec{j} and the vector potential \vec{A} have only ϕ -components, so that the equation to be solved is

$$\nabla \times \left(\frac{1}{\mu} \nabla \times A_\phi \vec{e}_\phi \right) + i\omega \sigma A_\phi \vec{e}_\phi = j_\phi \vec{e}_\phi. \quad (67.2)$$

The vector potential satisfies now automatically the Coulomb gauge. After the solution the heat generation in the conductors may be computed from

$$h = \frac{1}{2} \sigma \omega^2 |\vec{A}_0|^2.$$

In contrast to the stationary case where A_ϕ is real and the equation has only one unknown, in the harmonic case the equation has two unknowns — the in-phase and the out-of-phase component of the vector potential. The magnetic flux density may generally be calculated from the vector potential as a post-processing step. Both the vector potential and the magnetic flux density components are then provided. The variable names in the result file are magnetic vector potential and magnetic flux density i , with $i=1$ and $i=2$.

67.2.1 Boundary Conditions

For the magnetostatics equation one can apply either Dirichlet or homogeneous natural boundary conditions. In both cases, one must check that the computational domain is extended far enough to avoid numerical errors.

The Dirichlet boundary condition for A_ϕ is

$$A_\phi = A_\phi^b. \quad (67.3)$$

In practice, when the Dirichlet condition is used, one usually takes $A_\phi^b = 0$. If a Dirichlet condition is not specified, the homogeneous natural boundary condition is used.

67.3 Keywords

Simulation

Frequency Real

Frequency f if harmonic simulation is used.

Angular Frequency Real

Angular frequency $\omega = 2\pi f$ if harmonic simulation is used, alternative to the previous one.

Constants

Permeability of Vacuum Real $[4\pi 10^{-7}]$

Solver `solver id`

Note that all the keywords related to linear solver (starting with `Linear System`) may be used in this solver as well. They are defined elsewhere.

Equation `String [Static Magnetic Field]`

The name of the equation.

Variable `String [Aphi]`

The name of the variable.

Variable Dofs `Integer`

Number of dofs in the field, this should be one for the steady-state case and two for time-harmonic analysis.

Procedure `File ["StatMagSolve" "StatMagSolver"]`

The name of the file and subroutine.

Harmonic Simulation `Logical`

Assume time-harmonic simulation.

Calculate Magnetic Flux `Logical [True]`

By this flag the computation of the magnetic flux is activated. The default is `False`.

Calculate Magnetic Flux Abs `Logical [True]`

Sometimes it is useful to have the absolute magnetic flux available for nonlinear material laws. Then this flag can be turned on. The default is `False`.

Calculate Joule Heating `Logical [True]`

In large computations the automatic computation of the Joule heating may be turned off by this keyword. The default is `False`. The keyword is only applicable for the harmonic case. The computation results to two additional variables. `Joule Heating` gives the absolute heating and `Joule Field` the field that gives the heating when multiplied by the electric conductivity. This may be needed if the electric conductivity is discontinuous making also the heating power discontinuous.

Desired Heating Power `Real`

A constant that gives the desired total heating power in Watts. If the keyword is active, then the `Joule Heating` and `Joule Field` are multiplied by the ratio of the desired and computed heating powers.

Nonlinear System Convergence Tolerance `Real`

This keyword gives a criterion to terminate the nonlinear iteration after the relative change of the norm of the field variable between two consecutive iterations k is small enough

$$\|A_\phi^k - A_\phi^{k-1}\| < \epsilon \|A_\phi^k\|,$$

where ϵ is the value given with this keyword.

Nonlinear System Max Iterations `Integer`

The maximum number of nonlinear iterations the solver is allowed to do. If neither the material parameters nor the boundary conditions are functions of the solution, the problem is linear and this should be set to be 1.

Nonlinear System Relaxation Factor `Real`

Giving this keyword triggers the use of relaxation in the nonlinear equation solver. Using a factor below unity is sometimes required to achieve convergence of the nonlinear system. A factor above unity might speed up the convergence. Relaxed variable is defined as follows:

$$A_\phi' = \lambda A_\phi^k + (1 - \lambda) A_\phi^{k-1},$$

where λ is the factor given with this keyword. The default value for the relaxation factor is unity.

Equation `eq id`

The equation section is used to define a set of equations for a body or set of bodies:

Static Magnetic Field Logical

If set to True, solve the magnetostatics equation.

Body Force bf id

The body force section may be used to give additional force terms for the equations.

Current Density Real

Specifies the azimuthal component of the current density. It may be a positive or negative constant, or a function of a given variable.

Current Phase Angle Real

Specifies the phase angle of the current density in degrees. The default phase angle is zero. Applies only to the time-harmonic case.

Joule Heat Logical

If this flag is active, the heat equation will automatically include the computed Joule heating as a heat source. Then it is assumed that Joule heating field ϕ is named `Joule field`. If there is no heat equation, this flag has no effect.

Initial Condition ic id

The initial condition section may be used to set initial values for the field variables. The following variable is active:

Aphi Real

The azimuthal component of the magnetic vector potential.

Material mat id

The material section is used to give the material parameter values. Material parameter available for the magnetostatics equation are.

Relative Permeability Real

The relative magnetic permeability μ is set with this keyword, defining the material relation $\vec{B} = \mu_r \mu_0 \vec{H}$. By default the relative magnetic permeability is one, but it may also be set otherwise or be a function of a given variable, typically given by the relation $\mu_r = \mu_r(|\vec{B}|)$. The value of the magnetic flux density $|\vec{B}|$ is available by the variable named `Absolute Magnetic Flux`.

Electric Conductivity Real

The electric conductivity defines the relation $\vec{j} = \sigma \vec{E}$. Only isotropic case is possible. The parameter is needed only in the time-harmonic case.

Boundary Condition bc id

The boundary condition section holds the parameter values for various boundary condition types. A Dirichlet boundary condition may be set for the vector potential. The one related to the axially symmetric magnetostatics problem is

Aphi Real

The azimuthal component of the magnetic vector potential.

Model 68

Electrostatics of Moving Rigid Bodies

Module name: MovingElstatSolver
Module subroutines: MovingElstatSolver
Module authors: Peter Råback
Document authors: Peter Råback
Document created: 26.1.2006
Document edited: 17.5.2006

68.1 Introduction

This solver is tailored for solving electrostatic problems that occur in the movement of rigid bodies in respect to one another. Here the movement is assumed to be a combination of rotations and translations. We are mostly interested in lumped quantities. The most important quantity is the capacitance of the moving body at different positions. In addition the sensitivity of the capacitance and the moment of the electric force may be computed. The information is saved in a generic tabulated form and also as a lumped circuit model of Aplac.

For a more generic cases of the electrostatics and mesh adaptation the user is encouraged to use the existing separate solvers.

68.2 Electrostatics

Assuming a constant permittivity ε , absence of free charges, and non-conducting media the equation for the electrostatic potential ϕ yields,

$$-\varepsilon \nabla \cdot \nabla \phi = 0. \quad (68.1)$$

Obviously the constant multiplier may be dropped for convenience.

The energy of the electric field may be computed from

$$E = \frac{1}{2} \varepsilon \int_{\Omega} |\nabla \phi|^2 d\Omega. \quad (68.2)$$

If there is only one potential difference Φ present then the capacitance C may be computed from

$$C = \frac{2E}{\Phi^2}. \quad (68.3)$$

The electric force is calculated by integrating the electrostatic Maxwell stress tensor over the specified surface. Using the stress tensor $\bar{\bar{T}}$ the total force on the surface S can be expressed as

$$\vec{F} = \int_S \bar{\bar{T}} \cdot \vec{n} dS. \quad (68.4)$$

The components of the Maxwell stress tensor for linear medium are

$$T_{ij} = -D_i E_j + \frac{1}{2} \delta_{ij} \vec{D} \cdot \vec{E}, \quad (68.5)$$

where electric field \vec{E} and electric displacement field \vec{D} are obtained from

$$\vec{E} = -\nabla \phi, \quad (68.6)$$

and

$$\vec{D} = -\varepsilon \nabla \phi. \quad (68.7)$$

The moment around a given point \vec{r}_0 is given by,

$$\vec{F} = \int_S \vec{T} \cdot \vec{n} \times (\vec{r} - \vec{r}_0) dS. \quad (68.8)$$

One may get an secondary estimation for the capacitance from the integral of the surface charges.

$$C' = \frac{1}{\Phi} \int_S \vec{D} \cdot \vec{n} dS. \quad (68.9)$$

This estimate of capacitance has a much bigger numerical error than the one defined by the volume integral. However the estimate may be useful in evaluating the accuracy of the capacitance. The volume integral approaches the exact capacitance from above while the surface integral approaches it from below.

68.3 Mesh movement

The generic mesh movement of Elmer is based on a linear elasticity model. This is often an overkill since this makes the solution of the mesh movement computationally much more expensive than the solution of the potential equation. Therefore in the mesh movement it is assumed that the displacements of the main direction u_i , $i = 1, 2, 3$ are independent. Then the displacement of each directions is given by the Laplace equation

$$-\nabla \cdot \nabla u_i = 0. \quad (68.10)$$

The obvious boundary conditions for the displacements would be to fix the displacements at all walls. However, for large movement this would distort the mesh unnecessarily. Therefore the displacements are fixed only in the direction of the surface normal. Thus, if the normal is the main direction, the other two components are free to slide. This approach is unfortunately feasible only for rectangular geometries. Also the displacements of the mesh points at the outer boundaries will be fixed. An outer boundary is assumed to be moving if it is somewhere attached to the moving body, otherwise it is assumed to be at rest.

For the moving rigid body the displacements are given by three translational U_i and three rotational Φ_i degrees of freedom. Then the displacement at the moving wall yields

$$\vec{u} = \vec{U} + R(\vec{r} - \vec{r}_0), \quad (68.11)$$

where

$$R = \begin{pmatrix} 0 & \Phi_z & -\Phi_y \\ -\Phi_z & 0 & -\Phi_x \\ \Phi_y & \Phi_x & 0 \end{pmatrix} \quad (68.12)$$

It should be noted that if the movement of the frame is pure translational then there may be no need to solve the displacements in all directions since the Laplace has a non-zero solution only if some of the boundary conditions is non-zero.

68.4 Implimentation issues

For simple equations, such as the Laplace equation, a large part of the computational effort goes into the assembly of the linear equation. In this special case the same matrix may be assembled only once and used repetitively to solve the mesh adaption problem varying just the boundary conditions. Unfortunately, the potential equation must every time be reassembled as the coordinates change.

The purpose of the simulation is to get detailed information of the capacitance in respect with the rigid body movement. However, as there are six degrees of freedom making just 10 observations in each direction would result to 10^6 , that is a million, simulations. Therefore it is advisable to make the observations only to some predefined directions. For this purpose the user may give up to six basis $\vec{\eta}_i$ to define these directions. By default $\eta_{ij} = \delta_{ij}$.

For each basis $\vec{\eta}_i$ the user may also give the interval of the amplitude $[a_i, b_i]$ and the number of observation points N_i . Then the rigid body coordinates are

$$\vec{U} = \sum_{i=1}^6 \left(a_i + (n_i - 1) \frac{b_i - a_i}{N_i - 1} \right) \vec{\eta}_i. \quad (68.13)$$

68.5 Keywords

Constants

Permittivity Of Vacuum Real [8.8542e-12]

Solver solver id

Equation String MovingElstatSolver

Variable String Potential

This may be of any name as far as it is used consistently also elsewhere.

Variable DOFs Integer 1

Degrees of freedom for the potential.

Procedure File "MovingElstatSolver" "MovingElstatSolver"

Following are listed four keywords with default values for output control.

Moment About i Real

The center of coordinate system ($i = 1, 2, 3$) for the rigid body movement and for computing the moments.

Lumping Basis j(6) Real

The basis $\vec{\eta}_j$, $j = 1, 2, \dots, 6$.

Lumping Points j Integer

Number of observations for basis j .

Lumping Interval j(2) Real

The interval of amplitude for basis j .

Length Scale Real

The Aplac export assumes certain unit system. Therefore if the length unit of the mesh is not given in metres this value may be given to rescale the results appropriately.

Calculate Force Logical [True]

Whether to calculate and save the force lumped force.

Calculate Moment Logical [True]

Whether to calculate and save the force lumped moments..

Save Displacements Logical True

Whether to save the displacement field in the ElmerPost format.

Filename File

All the results are saved in the file given by this keyword. Additionally a suffix `.info` is given to file that explains what is being saved. Finally, if Aplac model is created, it is given the suffix `.apla`.

Boundary Condition bc id

Potential Real

Moving Boundary Logical

If this is true then displacements are fixed using the rigid body movement and potential is given value one.

Fixed Boundary Logical

If this is true then displacements are fixed to zero in normal direction and potential is given value zero.

Periodic BC Potential Logical

Periodic boundary conditions for the potential is activated by this keyword. Note that this affects only the potential solution. The displacements at the symmetric boundaries are fixed internally to zero.

Periodic BC Integer

The periodic counterpart of the current Boundary Conditions.

Periodic BC Translate(3) Real

This keyword is required for the older versions of Elmer code to give the translational vector of the periodicity.

Model 69

Fluidic Force

Module name: FluidicForce

Module subroutines: ForceCompute

Module authors: Juha Ruokolainen, Antti Pursula

Document authors: Antti Pursula

Document edited: Feb 28th 2005

69.1 Introduction

This module is used to calculate the force that a fluid flow induces on a surface. The fluidic force can be divided into two main components: force due to pressure and viscous drag force. The fluid can be compressible or incompressible and also non-Newtonian with the same limitations than there are in the Elmer Navier-Stokes Equation solver. The force calculation is based on a flow solution (velocity components and pressure) which has to present when calling the procedure. Also the torque with respect to a given point can be requested.

69.2 Theory

The force due to fluid is calculated as a product of the stress tensor and normal vector integrated over the surface

$$\vec{F} = \int_S \bar{\bar{\sigma}} \cdot \vec{n} dS. \quad (69.1)$$

The stress tensor is

$$\bar{\bar{\sigma}} = 2\mu\bar{\bar{\varepsilon}} - \frac{2}{3}\mu(\nabla \cdot \vec{u})\bar{\bar{I}} - p\bar{\bar{I}}, \quad (69.2)$$

where μ is the viscosity, \vec{u} is the velocity, p is the pressure, $\bar{\bar{I}}$ the unit tensor and $\bar{\bar{\varepsilon}}$ the linearized strain rate tensor, i.e.

$$\varepsilon_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right). \quad (69.3)$$

The torque about a point \vec{a} is given by

$$\vec{\tau} = (\vec{r} - \vec{a}) \times \vec{F}(\vec{r}), \quad (69.4)$$

where \vec{r} is the position vector.

69.3 Additional output

There is also a feature for saving the tangential component of the surface force i.e. the shear stress element-wise on the boundaries. The shear stress output is written on disk in a file which contains three columns: 1) the value of the shear stress, 2 and 3) the corresponding x and y coordinates. The shear stress is saved on all boundaries where fluidic force computation is requested. This feature is implemented only for 1D-boundaries of 2D-geometries.

69.4 Keywords

`Solver solver id`

`Equation String Fluidic Force`

`Procedure File "FluidicForce" "ForceCompute"`

`Calculate Viscous Force Logical [True]`

Setting this flag to false disables the viscous drag force, and only the surface integral of pressure is calculated.

`Sum Forces Logical [False]`

By default the solver calculates the fluidic force by boundaries. Setting this flag to True applies summing of each individual boundary force in to a resultant force which is the only force vector in output.

`Shear Stress Output Logical [False]`

Setting this flag to True activates writing shear stress values on disk.

`Shear Stress Output File String [shearstress.dat]`

Defines the name of the shear stress file.

`Velocity Field Name String`

The name of the velocity field variable. This keyword may be necessary if some other flow solver than the built-in Navier-Stokes solver of Elmer is used. Normally this keyword should be omitted.

`Material mat id`

`Viscosity Real`

`Boundary Condition bc id`

`Calculate Fluidic Force Logical [True]`

The fluidic force is calculated for the surfaces where this flag is set to true.

`Moment About(dim) Real`

Coordinates for the point on which the torque is returned.

Model 70

Electrostatic force

Module name: ElectricForce

Module subroutines: StatElecForce

Module authors: Antti Pursula

Document authors: Antti Pursula

Document edited: February 7th 2003

70.1 Introduction

This solver calculates the electrostatic force acting on a surface. The calculation is based on an electrostatic potential which can be solved by the electrostatic solver (see Model 15 of this Manual).

70.2 Theory

The force is calculated by integrating the electrostatic Maxwell stress tensor [1] over the specified surface. Using the stress tensor $\bar{\bar{T}}$ the total force on the surface S can be expressed as

$$\vec{F} = \int_S \bar{\bar{T}} \cdot d\vec{S}. \quad (70.1)$$

The components of the Maxwell stress tensor for linear medium are

$$T_{ij} = -D_i E_j + \frac{1}{2} \delta_{ij} \vec{D} \cdot \vec{E}, \quad (70.2)$$

where electric field \vec{E} and electric displacement field \vec{D} are obtained from the electric potential Φ

$$\vec{E} = -\nabla \Phi, \quad (70.3)$$

and

$$\vec{D} = -\varepsilon_0 \varepsilon_r \nabla \Phi, \quad (70.4)$$

where ε_0 is the permittivity of vacuum and ε_r is the relative permittivity of the material, which can be a tensor.

70.3 Keywords

Constants

Permittivity Of Vacuum Real [8.8542e-12]

Solver solver id

Equation String Electric Force

The name of the equation. Not necessary.

Procedure File "ElectricForce" "StatElecForce"

Exec Solver String After Timestep

Often it is not necessary to calculate force until solution is converged.

Material mat id

Relative Permittivity Real

Boundary Condition bc id

Calculate Electric Force Logical True

This keyword marks the boundaries where force is calculated.

Bibliography

[1] J. Vanderlinde. *Classical electromagnetic theory*. John Wiley & Sons, 1993.

Model 71

Linear Constraints

Module name: included in solver (SolverUtils)

Module subroutines: SolveWithLinearRestriction

Module authors: Mika Juntunen

Document authors: Mika Juntunen

Document edited: August 5th 2003

71.1 Introduction

This subroutine allows user to solve problems with linear constraints. Here constraints are forced with Lagrange multipliers. This method, however, does not always lead to a well-posed problem. Conditions that ensure a (unique) solution are excluded here, but the conditions are found in many books (check for example [1]).

71.2 Theory

The problem at hand is

$$\min_x x^T A x - x^T f \quad (71.1)$$

Let's assume that we can solve this. Now we also want that the solution solves the system $Bx = g$. This gives constraints to our solution. The rank of B should be less or equal to the rank of A . Loosely speaking, the number of rows in B should be less or equal to the number of rows in A . The method of Lagrange multipliers fixes these two equations together and gives a new functional to minimize.

$$\min_x x^T A x - x^T f + \lambda^T (Bx - g) \quad (71.2)$$

If A is symmetric, then simple variational approach leads to solving x out of system

$$\begin{pmatrix} A & B^T \\ B & 0 \end{pmatrix} \begin{pmatrix} x \\ \lambda \end{pmatrix} = \begin{pmatrix} f \\ g \end{pmatrix} \quad (71.3)$$

Symmetry of A is not always needed, but then more powerful methods have to be used to get to the above system.

71.3 Limitations

- **General usage of the subroutine**

This subroutine can not be used by just adding keywords to solver input file. You must somehow create the constraint matrix and then call for SolveWithLinearRestriction in your own subroutine or function. The reader is encouraged to check for details in ElmerTutorials.

- **EMatrix-field**

The EMatrix-field of the solved system matrix is used passing constraint matrix to SolveWithLinearRestriction. This will be a problem if some other function or subroutine tries to use the EMatrix-field. EMatrix-field of the constraint-matrix is internally used by SolveWithLinearRestriction and should therefor be left alone.

- **Exported Multipliers**

The length of the vector that holds the multipliers is limited to be a multiply of the number of nodes in mesh. This means that the vector usually has extra entries. These entries are set to zero. This leads to problems in extracting the correct values from the result file. Also post processing with ElmerPost is at least tricky.

- Parallel solving is not yet implemented.

71.4 Keywords

Solver `solver-id`

Export Lagrange Multiplier `Logical`

If the multiplier has some physical meaning, you can save it to result file and to post file. This feature has certain drawbacks, check subsection Limitations. Default is `False`.

Lagrange Multiplier Name `String`

The name you want to call the exported multipliers. This keyword has no meaning if the previous keyword is set to `False`. Default name is `LagrangeMultiplier`.

Bibliography

- [1] V. Girault and P.A. Raviart. *Finite element methods for Navier-Stokes equations*. Springer-Verlag, New York, Berlin, Heidelberg, 1986.

Model 72

Density Functional Theory

Module name: DFTSolver

Module subroutines: Poisson, WaveFunctionSolver, ChargeDensitySolver, xc

Module authors: Olli Mali, trad (xc)

Document authors: Olli Mali

Document created: 10.12.2006

Document edited: 18.12.2006

72.1 Introduction

This is an instructional text for using Elmer solvers I created for DFT calculations during the year 2006 while preparing my Master's Thesis [7]. These Solvers are rather experimental and I would not recommend their use for highly complicated problems. Nevertheless they provide nice backbone for creating own DFT-solvers with finite element method.

72.2 Theory

In DFT, Kohn-Sham equations [1, 2] play central role. They are set of highly nonlinear equations which define uniquely the exact ground state charge density. From charge density the total energy of the system in ground state can be calculated, which is unfortunately not implemented in present code.

The Kohn-Sham equations have a form

$$\begin{aligned} \left(-\frac{1}{2}\Delta + V_{EXT}(\mathbf{r}) + V_C[\rho(\mathbf{r})] + V_{XC}[\rho(\mathbf{r})] \right) \psi_k(\mathbf{r}) &= \varepsilon_k \psi_k(\mathbf{r}) \\ \rho(\mathbf{r}) &= \sum_{k=1}^N |\psi_k(\mathbf{r})|^2 \end{aligned} \quad (72.1)$$

where KS-orbitals $\psi_k(\mathbf{r})$ are normalized, $\int \psi_k(\mathbf{r})^2 d\mathbf{r} = 1$, for each $k = 1, 2, \dots, N$. V_{EXT} is the external potential caused by the nuclei, V_C is the non-interacting Coulomb potential and V_{XC} is the exchange correlation potential that includes all the complicated many body effects, at least approximates. Nice explanation from the widely used Local Density Approximation can be found from [3]. Nonlinearity occurs in eigenvalue problem, where the operator depends on the solution of the eigen problem.

Self-Consistent iteration

The equations (72.1) are solved with self-consistent iteration (fixed point iteration). In this iteration Coulomb and external potentials are solved from Poisson equation. The iteration steps are as follows:

1. Begin with previous or initial guess for charge density ρ^j

2. Solve new electric potential from Poisson equation,

$$-\Delta V^{j+1}(\mathbf{r}) = \frac{1}{4\pi} \rho^j(\mathbf{r}) - \sum_{i=1}^M Z_i \delta(\mathbf{r} - \mathbf{r}_i) \quad , \quad (72.2)$$

where δ refers to Dirac's delta distribution (point load).

3. Solve eigenvalue problem,

$$\left(-\frac{1}{2}\Delta + V^{j+1}(\mathbf{r}) + V_{xc}^{j+1}(\mathbf{r}) \right) \psi_k(\mathbf{r}) = \varepsilon_k \psi_k(\mathbf{r}) \quad , \quad (72.3)$$

where V_{xc}^{j+1} is calculated via some function W from point values of charge density ρ^j . $V_{xc}^{j+1}(\mathbf{r}) = W(\rho^j(\mathbf{r}))$.

4. Sum new charge density,

$$\rho^{j+1}(\mathbf{r}) = \sum_{k=1}^N w_k \psi_k(\mathbf{r})^2 \quad , \quad (72.4)$$

where the weight coefficients w_k depend on the numbers of electrons in orbitals. Extensive overview of calculation of molecular orbitals can be found from [4, 5].

The point load at the nuclei location requires, that *exactly at each nuclei there has to be a node* in the mesh. For the functionality of the solvers no other requirements exists for the mesh or domain.

Unfortunately convergence of this iteration procedure is not guaranteed. For simple atoms ($Z = 1, 2, 3, 4$) code converges within any tolerance limits but for more complicated molecules or atoms usually not. Sensible tolerances were found to be between 10^{-6} or 10^{-4} .

Boundary Conditions

In theory the zero level of the potential can be set arbitrarily and often in practice one uses condition $V(\mathbf{r}) \rightarrow 0$, when $|\mathbf{r}| \rightarrow \infty$. Of course in real calculations the domain Ω is finite and we set, $V(\mathbf{r}) = 0$ if $\mathbf{r} \in \partial\Omega$. One also assumes Ω to be large enough, so that charge density vanishes on the boundary, $\rho(\mathbf{r}) = 0$ if $\mathbf{r} \in \partial\Omega$, so we set $\psi_k(\mathbf{r}) = 0$ if $\mathbf{r} \in \partial\Omega$.

In Kohn-Sham -equations in order to obtain positive definite coefficient matrix on the left hand side of eigenvalue problem (72.1), one sets $V(\mathbf{r}) \rightarrow C$, when $|\mathbf{r}| \rightarrow \infty$. The constant C has to be large enough, so the eigenvalues are shifted positive. But too large value slows the convergence of the eigenvalue solver.

72.3 Keywords

From the structure of the self-consistent iteration it was natural to divide the solution procedure for three solvers, Poisson solver, eigensolver and charge density summation. For each solver some keywords to control the solution procedure were added.

Poisson Solver

Poisson Solver demands knowledge about the locations of the nuclei and their atomic numbers. There has to be nodes in the mesh at the nuclei locations, or else error will occur. Following example demonstrates how nuclei of the water molecule with two atoms of atomic numbers $Z = 1$ (Hydrogen) and single with $Z = 8$ (Oxygen) are set to the coordinates (0.0, 0.0, 0.0) (Oxygen) and (-1.43, 1.11, 0.0) and (1.43, 1.11, 0.0) (Hydrogens). The rows beginning with ! are comments.

```
!
! NOFnuclei is the number of nuclei in the structure.
!
```

```

NOFnuclei = Integer 3

!
! NucleiTable is an array of the form
! NucleiTable( NOFnuclei, 4 ) where each row
! includes the information of one nucleus.
! The columns are from left to right :
!
! atomic number, x-coordinate, y-coordinate and z-coordinate.
!

NucleiTable(3,4) = Real 8.0  0.0  0.0  0.0 \
                      1.0 -1.43 1.11 0.0 \
                      1.0  1.43 1.11 0.0

```

The self-consistent iteration requires heavy (under) relaxation to avoid divergence. Relaxation means linear mixing of present solution with previous one(s). It is possible to use *Guaranteed Reduction Pulay - method* [6, 7] where the mixing constants are calculated every time as a solution of a minimization problem, it's sensible to begin GR Pulay after some steps of linear mixing.

In following example the exponential relaxation scheme is changed to GR Pulay after 5 steps or if the mixing parameter exceeds value 0.5 . Use of constant mixing parameter instead of increasing one can be easily done by commenting out the first four uncommented lines and removing the comment sign ! from following two lines.

```

!
! Select the relaxation method used, possibilities are
! constant mixing parameter a(k) = A or varying parameter
! with scheme a(k) = C + 1- A * Exp( B * k )
!

Relaxation Method = String "Exponential mixing"
Relaxation Parameter A = Real 1.0
Relaxation Parameter B = Real 0.05
Relaxation Parameter C = Real 0.005

! Relaxation Method = String "Constant mixing"
! Relaxation Parameter A = Real 0.01

Start GRPulay after iterations = Integer 5
Start GRPulay if relaxation factor is more than = Real 0.5

```

Eigenproblem Solver

Eigenproblem solver demands knowledge about the type of exchange correlation approximation used. Namely the expression of W in third self-consistent iteration step. In module `xc.f90` there are several different formulae for LDA approximations. Some of them include spin directions and are to be used with different solver composition where KS-orbitals for up- and down-spins are calculated separately.

```

!
! Choose the type of the XC Potential, possible choices are:

```

```
! "None"
! "Perdew-Zunger"
! "Von Barth-Hedin"
! "Gunnarsson-Lundqvist"
! "Perdew-Wang"
!
```

```
XC Potential type = String "Perdew-Zunger"
```

Charge Density Solver

Charge density solver demands knowledge about the number of KS-orbitals to be summed and the weights of each orbital. These are the N and w_k 's in fourth self-consistent iteration step. In following example one sets $N = 5$ and $w_k = 2$, for all $k = 1, \dots, 5$.

```
! Define the number of eigenmodes included on the
! calculation of charge density. Set weights for the
! eigen states. By default they are all 1.
```

```
Number of Eigenmodes Included = Integer 5
Weights of Eigen States(5,1) = Real 2.0 2.0 2.0 2.0 2.0
```

Weights of the Eigen States table has to be size $(N, 1)$. Naturally N has to be equal or less for the number of eigenstates to be solved in eigenvalue solver.

Bibliography

- [1] P. Hohenberg, W Kohn *Inhomogeneous Electron Gas* Physical Review, Volume 136, Number 3B, 9 November 1964
- [2] W. Kohn, L. J. Sham, *Self-Consistent Equations Including Exchange and Correlation Effects* Physical Review, Volume 140, Number 4A, 15 November 1965
- [3] M. P. Das, *Density Functional Theory: Many-Body Effects Without Tears* Proceedings of the Miniworkshop on "Methods of Electronic Structure Calculations", ICTP, Trieste, Italy 10 August - 4 September 1992
- [4] Peter Atkins, Ronald Frieman, *Molecular Quantum Mechanics* Oxford University Press Inc., Fourth edition 2005
- [5] Andrew R. Leach, *Molecular Modelling*, Pearson Education Limited, Second edition 2001
- [6] D. R. Bowler, M. J. Gillan, *An efficient and robust technique for achieving self consistency in electronic structure calculations* Chemical Physics Letters 325 sivut 473-476 (2000)
- [7] O. Mali, *Kohn-Sham -yhtälöiden ratkaiseminen elementtimenetelmällä*, Diplomityö, 19. syyskuuta 2006

Model 73

Parallel I/O using HDF5 library

Module name: XdmfWriter
Module subroutines: XdmfWriter
Module authors: Mikko Lyly
Document authors: Mikko Lyly
Document created: 11.02.2011
Document edited: 20.02.2011

73.1 Introduction

This subroutine is intended for saving parallel results in Xdmf/HDF5 format. The advantages of the Xdmf/HDF5-format over the native parallel ep-format are the following:

- All results are stored in only two files (results.xmf and results.h5)
- The results are stored in a binary format with reduced storage requirements
- The results can be visualized on the fly during the solution

The result files written by XdmfWriter can be opened and visualized e.g. with Paraview.

At the moment, the module is available for the parallel version of Elmer only. Because of this, it has been isolated from the main build system. The source code for XdmfWriter can be found from the source tree in `misc/xdmf` and it should be compiled by the user as follows:

```
elmerf90 -I$HDF5/include -L$HDF5/lib -o XdmfWriter XdmfWriter.f90 -lhdf5_fortran -lhdf5
```

The environment variable `$HDF5` defines the installation directory for the HDF5-library.

73.2 Keywords

`Solver` `solver id`

`Equation` `String` `"Xdmf"`

The name of the equation.

`Procedure` `File` `"XdmfWriter"` `"XdmfWriter"`

The name of the file and subroutine.

`Base File Name` `String`

Specifies the base file name of the output files. The default is `results`.

`Single Precision` `Logical`

This keyword specifies the output precision (4 byte single precision floating point numbers vs. 8 byte double precision floating point numbers). The default is `false`.

The following keywords define the scalar and vector fields to be saved:

```
Scalar Field i String
    The scalar fields to be saved, for example Pressure.
```

```
Vector Field i String
    The vector fields to be saved, for example Velocity.
```

The number *i* must be in the range 1...1000.

73.3 Example

The following SIF-block saves results in Xdmf/HDF5-format for the Navier-Stokes equations:

```
Solver 1
  Equation = String "Xdmf"
  Procedure = File "XdmfWriter" "XdmfWriter"
  Base File Name = String "MyResults"
  Single Precision = Logical True
  Scalar Field 1 = String "Pressure"
  Vector Field 1 = String "Velocity"
End
```

Index

- accumulation, 182
- Active Coordinate, 217, 284, 290
- Active EigenModes, 259
- Additional Info, 235
- Advection Diffusion Equation Varname, 35
- AdvectionDiffusionSolver, 32
- AdvectionReactionSolver, 38
- ALE, 14, 25
- ALE Formulation, 184
- Angular Frequency, 84, 92, 119, 120, 129, 130, 151, 296, 303
- Angular Velocity, 28
- Angular Velocity i, 118
- Aphi, 305
- Applied Magnetic Field i, 29, 139
- Apply Dirichlet, 184
- Apply Limiter, 17, 45
- Arbitrary Lagrangian-Eulerian, 14, 25
- Artery Outlet Solver, 300
- Artery Poisson Ratio, 300
- Artery Radius, 300
- Artery Wall Thickness, 300
- Artery Wall Youngs Modulus, 300
- ArteryOutlet, 298
- Artificial Compressibility, 240, 241
- artificial compressibility, 237
- ArtificialCompressibility, 237
- Ascii Output, 259
- Automated Source Projection BCs, 117
- AV, 118
- AV {e} j, 118
- Average Within Materials, 135, 150, 225
- Az, 134
- Base File Name, 320
- BC Id Offset, 260
- BDF Order, 16, 34, 39
- BeamSolver3D, 78
- Before Linsolve, 235
- BEM, 292, 295
- Best File, 244
- Binary Output, 259
- bisection, 243
- Block Diagonal A, 167
- Block Preconditioning, 94, 167, 173
- Body, 191, 196, 234, 261
- Body Force, 28, 46, 53, 61, 71, 76, 81, 85, 97, 101, 105, 109, 118, 119, 125, 130, 134, 135, 138, 147, 168, 173, 178, 185, 207, 215, 291, 294, 305
- Body Force i, 101, 173
- Body Force k, 81
- Body Force Parameters, 257
- Body Forces, 19, 35, 40
- Body ID, 185
- Body Id, 192, 294, 297
- Boltzmann Constant, 146
- Boltzmann distribution, 144
- Bottom Surface, 218
- Bottom Surface Level, 218
- Bottom Surface Variable Name, 218
- Boundary Condition, 20, 30, 36, 41, 46, 54, 62, 71, 76, 81, 85, 93, 98, 106, 109, 118, 120, 130, 134, 139, 142, 147, 157, 161, 168, 173, 181, 185, 188, 191, 196, 204, 207, 210, 213, 216, 218, 232, 240, 251, 254, 261, 263, 280, 291, 294, 297, 300, 305, 309, 311, 313
- boundary element method, 292, 295
- boundary integral, 250
- Boundary Layer Thickness, 31
- Boussinesq, 28
- Boussinesq approximation, 24
- Box Contact Directions, 202
- Box Particle Collision, 202
- Box Particle Contact, 202
- Box Particle Periodic, 202
- Box Periodic Directions, 202
- BSolver, 132
- Bubbles, 17, 35, 84, 290
- Bubbles in Global System, 92
- Bulk Modulus, 46
- Bulk Viscosity, 93, 101
- Calculate Acoustic Impedance, 94
- Calculate Capacitance Matrix, 105
- Calculate Coil Current, 156
- Calculate Current Density, 120
- Calculate Div of Poynting Vector, 131
- Calculate Electric Energy, 105, 146
- Calculate Electric Field, 105, 120, 146
- Calculate Electric field, 131

- Calculate Electric Flux, 105, 146
- Calculate Electric Force, 313
- Calculate Elemental Fields, 121, 130, 151
- Calculate Energy Functional, 131
- Calculate Fluidic Force, 311
- Calculate Flux, 46, 225
- Calculate Flux Abs, 225
- Calculate Flux Dim, 47
- Calculate Flux Magnitude, 225
- Calculate Force, 46, 308
- Calculate Force Dim, 47
- Calculate Grad, 225
- Calculate Grad Abs, 225
- Calculate Grad Magnitude, 225
- Calculate Harmonic Loss, 120
- Calculate Heating, 46
- Calculate Joule Heating, 109, 120, 135, 304
- Calculate Magnetic Field Strength, 120, 130
- Calculate Magnetic Flux, 304
- Calculate Magnetic Flux Abs, 304
- Calculate Magnetic Flux Density, 130
- Calculate Magnetic Force, 121
- Calculate Magnetic Torque, 121
- Calculate Matrix Suction, 290
- Calculate Maxwell Stress, 120
- Calculate Mesh Velocity, 215
- Calculate Moment, 308
- Calculate Nodal Fields, 121
- Calculate Nodal Heating, 109, 121
- Calculate Nodal Losses, 151
- Calculate PAngle, 61
- Calculate Pangle, 52
- Calculate Poynting Vector, 130
- Calculate Principal, 52, 61
- Calculate Stesses, 61
- Calculate Strains, 52, 61
- Calculate Stresses, 52
- Calculate Surface Charge, 105
- Calculate Viscous Force, 311
- Calculate Volume Current, 109
- Capacitance Bodies, 105
- Capacitance Body, 106
- capacitance matrix, 104
- Capacitance Matrix Filename, 105
- Cfix diffusion, 156
- Charge Density, 105, 147
- Charge Number, 147
- ChargeDensitySolver, 316
- Check for Duplicates, 263
- CircuitsAndDynamics, 122
- CircuitsAndDynamicsHarmonic, 122
- CircuitsOutput, 122
- coating, 186
- Coefficient i , 250
- Coil Anisotropic, 156
- Coil Bandwidth, 156
- Coil Center(3), 156
- Coil Closed, 156
- Coil Conductivity Fix, 156
- Coil Cross Section, 156
- Coil End, 157
- Coil Normal(3), 156
- Coil Start, 157
- Coil Type, 124
- CoilSolver, 153
- CoilSolvrr, 153
- Component, 121, 124
- Compressibility Model, 36, 46, 240
- CompressibilityScale, 237
- CompressibleNS, 99
- Compressibility Model, 20, 29
- Concentration Units, 35
- Conserve Volume, 180
- Conserve Volume Relaxation, 180
- Constant Bulk Matrix, 228, 229
- Constant Bulk System, 52
- Constant Weights, 105, 109, 146
- Constant-Viscosity Start, 166
- Constants, 15, 26, 105, 116, 128, 133, 142, 146, 160, 303, 308, 312
- continuity equation, 238
- Continuous Reading, 271
- Convection, 18, 28, 35, 40, 185
- Convection Velocity i , 20, 36, 41, 85, 191
- Convective, 167, 172
- Convert From Equation Name, 283
- Convert From Variable, 283
- Coordinate Condition Variable Name, 202
- Coordinate Initialization Method, 200, 206
- Coordinate System, 15, 34, 39, 60, 70, 80, 92, 100, 222
- Coordinate Transformation, 274
- Correct Surface, 218
- Correct Surface Mask, 218
- Cosine Series i , 278
- Cost Function Absolute, 244
- Cost Function Index, 249
- Cost Function Maximize, 244
- Cost Function Name, 244
- Cost Function Target, 244
- Create Histogram, 286
- Critical Shear Rate, 29
- Cross Section Area, 80
- Cumulative Displacements, 212, 215
- Current Control, 109
- Current Density, 109, 119, 134, 305
- current density, 107
- Current Density i , 130

- Current Density BC, 109, 157
- Current Density i , 118
- Current Density Im, 135
- Current Density im i , 130
- Current Density Im i , 120
- Current Phase Angle, 305
- Current Source, 109
- curvature, 178
- Curvature Coefficient, 180
- Curvature Diffusion, 180
- Damping, 55, 76
- Darcy's law, 25
- DataToFieldSolver, 267, 279
- Decay Time i , 278
- Decay Timestep i , 278
- Default BC Id, 259
- Default Body Id, 259
- Deflection i , 76
- Delete Wall Particles, 201
- delta function, 177
- Density, 19, 28, 36, 53, 60, 70, 76, 80, 85, 161, 166, 172, 191, 196, 300
- Desired Coil Current, 156
- Desired Current Density, 156
- Desired Heating Power, 135, 304
- DFTSolver, 316
- dielectric layer, 142
- Diffusion Coefficient, 268, 280
- Diffusivity Name, 268
- Dirichlet boundary condition, 15, 23, 33, 39, 133, 137, 303
- Discontinuous Bodies, 260
- Discontinuous Galerkin, 38, 40, 135, 150, 225, 260
- Displace Mesh, 52, 70
- Displaced Shape, 241
- Displacement i , 53, 54, 300
- Displacement Mode, 218
- Displacement Variable Eigenmode, 84
- Displacement Variable Frequency, 85
- Displacement Variable Name, 84, 241
- Div Discretization, 27
- DivergenceSolver, 229
- DNU i , 71
- domain integral, 250
- Dot Product Tolerance, 218, 285
- drawing, 186
- Dx Format, 258
- E im {e}, 130
- E im {f}, 130
- E re {e}, 130
- E re {f}, 130
- Echo Values, 249
- effective parameters, 74
- Eigen Analysis, 52, 75, 235, 259
- Eigen System Damped, 235
- Eigen System Use Identity, 235
- Eigen System Values, 52, 76, 235
- ElasticSolve, 56
- ElasticSolver, 56
- Electric Conductivity, 109, 116, 133, 139, 305
- Electric Conductivity Im, 119
- Electric Conductivity im, 134
- Electric Current Density, 119
- Electric Current Density Im, 120
- Electric Field , 160
- Electric Flux, 106
- Electric Flux BC, 147
- Electric Infinity BC, 106
- Electric Potential, 118
- Electric Robin Coefficient, 130
- Electric Robin Coefficient im, 130
- Electric Transfer Coefficient, 119
- Electric Transfer Coefficient Im, 120
- ElectricForce, 312
- Electrode Area, 125
- Electrode Potential, 106
- Electrokinetics, 159
- electrokinetics, 26
- Element, 92, 100, 117, 167
- ElementStats, 286
- Elmer2OpenFoamIO, 267
- Emissivity, 20, 21
- Enable Scaling, 275
- Energy Accommodation Coefficient, 94
- energy conservation, 13
- energy method, 74
- Enforce Positive Magnitude, 225
- Enthalpy, 19
- EO Mobility, 161
- Equation, 16, 18, 26, 27, 34, 35, 40, 45, 46, 52, 53, 60, 61, 70, 75, 81, 84, 85, 92, 97, 100, 105, 109, 117, 119, 120, 124, 125, 129, 130, 134, 135, 137, 138, 146, 150, 155, 160, 166, 172, 178–180, 184, 187, 190, 195, 200, 206, 210, 215, 217, 222, 225, 228, 229, 231, 234, 240, 241, 244, 258, 262, 265, 268, 270, 272, 274, 280, 283, 284, 286, 290, 291, 293, 296, 300, 304, 308, 311, 313, 320
- Equilibrium Density, 93, 100
- Equilibrium Temperature, 93, 100
- Eulerian, 176
- Exact Coordinates, 249
- Excess Pressure, 98
- Exec Solver, 313
- Export Lagrange Multiplier, 125, 315
- Exported Variable 1, 40, 137, 179, 184, 293, 296

- Exported Variable 1 DOFs, 184, 293, 296
- Extend Elastic Layers, 235
- Extend Elastic Region, 235
- External Concentration, 36
- External Pressure, 30
- External Temperature, 20
- Extract Interval, 179
- FieldName, 281
- FieldName Continue, 281
- File Append, 179, 249, 254
- Fileindex Offset, 263
- Filename, 179, 248, 254, 268, 274, 309
- Filename Numbering, 249
- Filename Particle Numbering, 204, 263
- Filename Prefix, 203, 262
- Filename Timestep Numbering, 204, 263
- FilmPressure, 46
- FilmPressure Lower Limit, 46
- filtering, 276
- FilterTimeSeries, 276
- FindOptimum, 243
- Fit Coefficient, 268
- Fix Displacements, 52
- Fix Input Current Density, 117
- Fixed Boundary, 216, 309
- Fixed Parameter i, 244
- Flow Admittance, 46
- Flow BodyForce i, 28, 168
- Flow Force BC, 30
- Flow Interface, 85
- Flow Model, 26
- FlowSolve, 22
- FlowSolver, 22
- Fluid Coupling With Boundary, 300
- fluid-structure interaction, 237
- FluidicForce, 310
- Flux, 294
- Flux 1, 297
- Flux 2, 297
- Flux Coefficient, 225, 232, 254
- Flux Variable, 232, 254
- FluxSolver, 224
- Force BC, 240
- Force i, 54
- Force i Im, 54
- Fourier Integrate Cycles, 151
- Fourier Loss Filename, 151
- Fourier series, 276
- Fourier Series Components, 151
- Fourier Series Output, 151
- Fourier Start Cycles, 151
- Fourier Start Time, 151
- Fourier Start Timestep, 151
- FourierLossSolver, 148
- Free Moving, 30
- Free Surface, 30
- free surface, 176
- Free Surface Bottom, 188
- Free Surface Number, 188
- Free Surface Reduced, 188
- FreeSurfaceReduced, 186
- FreeSurfaceSolver, 182
- Frequency, 52, 84, 92, 151, 303
- Frequency i, 278
- Friction Heat, 19
- FSI BC, 62
- Gap Height, 45, 143
- Gebhardt factors, 15, 17
- Gebhardt Factors Fixed After Iterations, 18
- Gebhardt Factors Fixed Tolerance, 18
- Gebhardt Factors Solver Full, 18
- Gebhardt Factors Solver Iterative, 18
- Genetic algorithm, 243
- Geometric Stiffness, 52
- Geometry Id, 261
- GiD, 258
- Gid Format, 258
- Gmsh, 258
- Gmsh Format, 258
- Gradp Discretization, 27
- Grashof convection, 23
- Gravity, 26
- Green's function, 296
- Grid dx, 263
- Grid nx, 263
- Grid Origin i, 263
- GridDataReader, 273
- Guess File, 244
- H-B Curve, 133
- Harmonic Analysis, 52
- Harmonic Loss Coefficient i, 151
- Harmonic Loss Field Exponent(K), 151
- Harmonic Loss Frequency Exponent(K), 151
- Harmonic Loss Linear Coefficient, 121
- Harmonic Loss Quadratic Coefficient, 121
- Harmonic Simulation, 304
- Heat Capacity, 20, 36
- Heat Conductivity, 20, 93, 101, 191
- Heat Equation, 18, 138
- Heat Expansion Coefficient, 29, 54
- Heat Flux, 20
- Heat Flux BC, 20
- Heat Source, 19
- Heat Transfer Coefficient, 20
- HeatSolve, 13
- HeatSolver, 13
- Heaviside function, 177

- Helmholtz, 85
- HelmholtzBEM, 296
- HelmholtzBEMSolver, 295
- HelmholtzSolver, 83
- Histogram Intervals, 286
- History File, 244
- Hole Correction, 76
- Hole Depth, 143
- Hole Fraction, 76, 143
- Hole Size, 76, 143
- Hole Type, 143
- Hydraulic Conductivity, 290

- ILU Order for Schur Complement, 95
- ILU Order for Velocities, 95
- Im Body Force i, 93
- Im Heat Source, 93
- Im Reference Wall Velocity i, 94
- Im Specific Acoustic Impedance, 93
- Im Specific Thermal Impedance, 94
- Im Surface Traction i, 93
- Im Temperature, 93
- Im Velocity i, 93
- Impedance Target Boundary, 94
- Implicit Gebhardt Factor Fraction, 17
- Impose Body Force Current, 120
- induction equation, 136
- Inertial Bodyforce j, 61
- Inexact Integration, 150
- Infinity BC, 134
- inflow boundaries, 39
- Initial Condition, 19, 28, 36, 40, 53, 85, 97, 139, 185, 299, 305
- Initial Coordinate, 200
- Initial Coordinate Search, 201
- Initial Parameter i, 244
- Initial Sphere Center, 200
- Initial Sphere Radius, 200
- Initial Velocity, 201
- Initial Velocity Amplitude, 201
- Initial Velocity Time, 201
- Initial Volume, 180
- Initialization Condition Variable, 200
- Initialization Mask Variable, 200, 206
- Initialize State Variables, 61
- integral equation, 292, 295
- Integral Heat Source, 19
- Integration Points At Radius, 283
- Internal history, 244
- Internal Mesh Movement, 196
- Interpolation Multiplier, 275
- Interpolation Offset, 275
- Inverse Permeability, 129
- Inverse Permeability im, 129
- Inverse Relative Permeability, 129
- Inverse Relative Permeability im, 129
- Ion Density, 147
- Is Time Counter, 274
- Is Time Index, 274
- Isosurface, 265
- Isosurface Value i, 254, 285
- Isosurface Values, 265
- Isosurface Variable, 265
- Isosurface Variable i, 254, 285

- Jfix, 118
- Joule Heat, 19, 109, 138, 305
- Joule heating, 14, 108

- KE C1, 29
- KE C2, 29
- KE Cmu, 29
- KE SigmaE, 29
- KE SigmaK, 29
- Kinetic Energy, 28
- Kinetic Energy Dissipation, 28
- Knudsen number, 43

- Lagrange Gauge Penalization Coefficient, 117
- Lagrange Multiplier Name, 315
- Lagrange multipliers, 314
- Lagrangian, 176
- Large Deflection, 70
- Latent Heat, 191, 196
- Latent heat, 189
- Layer Charge Density, 106
- Layer Index i, 285
- Layer Permittivity, 143
- Layer Relative Permittivity, 106
- Layer Thickness, 106, 143
- Length Scale, 308
- level-set method, 176
- LevelSet, 176
- LevelSet Bandwidth, 180
- Levelset bandwidth, 181
- LevelSet Convect, 179
- LevelSet Courant Number, 181
- Levelset Curvature BC, 181
- LevelSet Flux, 178
- LevelSet Timestep Directional, 181
- LevelSet Variable, 179, 180
- LevelSet Velocity 1, 179
- LevelSet Velocity 2, 179
- LevelSet Velocity i, 178
- LevelSetCurvature, 176
- LevelSetDistance, 176
- LevelSetIntegrate, 176
- LevelSetSolver, 176
- LevelSetTimestep, 176
- Limit Solution, 40

- Linear Constraints, 314
- Linear System Adaptive Tolerance, 167
- Linear System Base Tolerance, 168
- Linear System Convergence Tolerance, 71, 95, 167, 173, 226, 228, 230
- Linear System GCR Restart, 168
- Linear System Iterative Method, 118, 226, 228, 230
- Linear System Max Iterations, 95, 168, 173, 226, 228, 230
- Linear System Preconditioning, 118, 226, 228, 230
- Linear System Preconditioning Damp Coefficient, 129
- Linear System Preconditioning Damp Coefficient im, 129
- Linear System Refactorize, 118
- Linear System Relative Tolerance, 167
- Linear System Solver, 226, 228, 230
- Liquid, 191, 196
- Lorentz Force, 28, 138
- Lorentz force, 23
- Lorentz Velocity i, 118
- Lumped Acceleration After Iterations, 195
- Lumped Acceleration Limit, 195
- Lumped Acceleration Mode, 195
- Lumping Basis j(6), 308
- Lumping Interval j(2), 308
- Lumping Points j, 308

- Magnetic Bodyforce i, 138
- Magnetic Boundary Load i, 130
- Magnetic Boundary Load i im, 130
- Magnetic Field i, 139
- Magnetic Field Strength i, 118
- Magnetic Field Strength Im i, 120
- Magnetic Flux Density {n}, 119
- Magnetic Flux Density i, 119
- Magnetic Flux Density Im {n}, 120
- Magnetic Flux Density Im i, 120
- Magnetic Induction, 28, 138
- Magnetic Permeability, 139
- Magnetic Transfer Coefficient, 119
- Magnetic Transfer Coefficient Im, 120
- MagneticSolve, 136
- MagneticSolver, 136
- Magnetization i, 118, 133
- Magnetization i Im, 134
- Magnetization Im i, 120
- MagnetoDynamics, 110
- MagnetoDynamics2D, 132
- MagnetoDynamics2DHarmonic, 132
- MagnetoDynamicsCalcFields, 110
- magnetohydrodynamics, 136
- magnetostatics, 302
- Mask Condition, 260
- Mask Diffusion, 280
- Mask Name, 260, 263
- Mask Name i, 249
- Mask Variable, 260, 280
- mass conservation, 186
- Mass Transfer Coefficient, 36
- Master Bodies, 124
- Material, 19, 28, 36, 41, 45, 53, 60, 70, 76, 80, 85, 92, 97, 100, 105, 109, 116, 119, 121, 128, 133, 134, 139, 146, 151, 161, 166, 172, 178, 179, 181, 191, 196, 210, 213, 240, 241, 290, 300, 305, 311, 313
- Material Constants, 60
- Material Coordinates Unit Vector 1(3), 54
- Material Coordinates Unit Vector 2(3), 54
- Material Coordinates Unit Vector 3(3), 54
- Matrix Topology Fixed, 17
- Max Characteristic Speed, 201
- Max Coordinate i, 263
- Max Cumulative Time, 201
- Max Initial Coordinate i, 200
- Max Inner GCR Iterations, 94
- Max Mask Value, 280
- Max Outer Iterations, 94
- Max Output Level, 125
- Max Parameter i, 244
- Max Relative Radius, 283
- Max Step Size, 245
- Max Timestep Intervals, 201, 206
- Max Timestep Size, 201
- Maximum Displacement, 184
- Maxwell stress tensor, 306, 312
- Maxwell's equations, 103
- Mean Free Path, 46
- Melting Point, 196
- Melting point, 189
- Mesh Coefficient i, 213
- Mesh Deform i, 213
- Mesh Displace i, 216
- Mesh Force i, 213
- Mesh Matrix(dim,dim), 216
- Mesh Normal Force, 213
- Mesh Origin, 216
- Mesh Penalty Factor, 213
- Mesh Relax, 216
- Mesh Relax Normalize, 215
- Mesh Relax Source, 216
- Mesh Rotate, 215
- Mesh Rotation Axis Order(dim), 215
- Mesh Scale, 216
- Mesh Translate, 215
- Mesh Update, 210
- Mesh Update i, 185, 210
- Mesh Update Variable, 218
- Mesh Velocity 1, 54
- Mesh Velocity 2, 54

- Mesh Velocity 3, 54
- Mesh Velocity First Zero, 218
- Mesh Velocity Variable, 218
- MeshSolver, 209
- MHD Velocity i, 139
- Mid Surface, 218, 219
- Min Coordinate i, 263
- Min Initial Coordinate i, 200
- Min Mask Value, 280
- Min Parameter i, 244
- Min Timestep Size, 201
- Minimum Gebhardt Factor, 17
- Minimum Height, 218
- Minimum Hits At Radius, 283
- Minimum View Factor, 17
- Mixed Formulation, 61
- Model Lumping, 52
- Model Lumping Boundary, 55
- Model Lumping Filename, 52
- Moment About i, 308
- Moment About(dim), 311
- Momentum Accommodation Coefficient, 94
- Mortar BC, 134
- Moving Boundary, 216, 309
- Moving Mesh, 213, 249
- Moving Wall, 204
- MovingElstatSolver, 306
- MyMask, 263

- Name, 60
- Narrow Band, 179
- Narrow Interface, 157
- natural convection, 23
- Navier-Stokes, 28, 138
- Navier-Stokes equation, 22
- Nelder-Mead, 243
- Neo-Hookean Material, 61
- Neumann boundary condition, 15
- Newmark Beta, 16, 34, 39
- Newton iteration, 24
- Newtonian, 22
- No Matrix, 124
- Nodal Penalty Factor, 213
- Nominal Potential Difference, 106
- Nominal Shear Rate, 29
- non-Newtonian, 22
- Nonlinear Iteration Method, 173
- Nonlinear System Convergence Tolerance, 16, 27, 34, 40, 45, 70, 134, 138, 146, 167, 173, 184, 188, 191, 290, 304
- Nonlinear System Max Iterations, 16, 27, 35, 40, 45, 134, 138, 146, 156, 167, 173, 191, 290, 304
- Nonlinear System Newton After Iterations, 16, 27, 146, 167, 173, 195
- Nonlinear System Newton After Tolerance, 16, 27, 146, 167, 173, 195
- Nonlinear System Relaxation Factor, 16, 27, 35, 138, 173, 187, 195, 240, 290, 304
- NonphysicalMeshSolver, 212
- Norm Variable Index, 207
- Normal Force, 54
- Normal Force Im, 55
- Normal Pressure, 71
- Normal Surface Traction, 62, 168
- Normal Tangential Velocity, 161
- Normal Target Body, 37, 294, 297
- Normal Variable, 191
- Normal Velocity, 45
- Normal-Tangential Displacement, 54
- Normal-Tangential Velocity, 30
- Normalize by Given Weight, 280
- Normalize by Nodal Weight, 280
- Normalize Coil Current, 156
- Number Of EigenModes, 259
- Number of Material Constants, 60
- Number of Particles, 200
- Number of State Variables, 60

- Ohm's law, 107
- Open DX, 258
- Open Side, 46
- OpenFoam Directory, 268
- OpenFoam Field, 268
- OpenFoam File, 268
- OpenFoam Mesh i, 268
- OpenFoam Timestep, 268
- OpenFOAM2ElmerFit, 267
- OpenFOAM2ElmerIO, 267
- Operator i, 207, 250, 277, 285
- Optimal Finish, 244
- Optimal Restart, 244
- Optimization, 243
- Optimization Method, 244
- Optimize Matrix Structure, 235
- orthotropic, 74
- Outflow Boundary, 98
- Outflow boundary, 173
- OutletCompute, 298
- Output Directory, 248, 254, 265
- Output File Name, 258
- Output Format, 203, 258, 262
- Output Interval, 203
- Output Node Types, 235

- P2-P1 Approximation, 167
- Parallel Operator i, 250
- Parallel Reduce, 249
- ParallelProjectToPlane, 282
- Parameter i, 256, 257

- Particle Accumulation, 204
- Particle Accumulation Max Shear, 204
- Particle Accumulation Max Speed, 204
- Particle Accurate At Face, 206
- Particle Bounciness, 203
- Particle Cell Fraction, 200
- Particle Cell Radius, 200
- Particle Charge, 203
- Particle Damping, 202
- Particle Decay Distance, 203
- Particle Decay Time, 203
- Particle Distance Integral Source, 207
- Particle Drag Coefficient, 203
- Particle Element Fraction, 200
- Particle Gravity, 202
- Particle Info, 204, 207
- Particle Lift, 202
- Particle Mass, 202
- Particle Node Fraction, 200, 206
- Particle Particle Collision, 202
- Particle Particle Contact, 202
- Particle Radius, 202
- Particle Release Fraction, 201
- Particle Release Number, 201
- Particle Save Fraction, 204
- Particle Spring, 203
- Particle Time Integral Source, 207
- Particle To Field, 203
- Particle To Field Mode, 203
- Particle Trace, 204
- Particle Wall, 207
- ParticleAdvector, 205
- ParticleDynamics, 197
- Partition Numbering, 249
- Passive OpenFOAM Coordinate, 268
- Passive Steps, 196
- perforated plate, 74
- Perform Mapping, 187
- Perfusion Density, 19
- Perfusion Heat Capacity, 19
- Perfusion Rate, 19
- Perfusion Reference Temperature, 19
- Periodic BC, 309
- Periodic BC Potential, 309
- Periodic BC Translate(3), 309
- Permeability, 117, 133
- Permeability of Vacuum, 116, 128, 133, 303
- Permittivity Of Vacuum, 105, 142, 146, 160, 308, 312
- Permittivity of Vacuum, 128
- Phase Change, 196
- Phase Change Model, 18
- Phase Change Side, 192
- Phase Change Variable, 190, 195
- Phase Velocity i, 196
- PhaseChangeSolve, 189, 193
- Physical Units, 36, 37
- Picard iteration, 24
- Plane Permutation, 283
- Plane Stress, 53, 60
- Poisson, 316
- Poisson Boltzmann Alpha, 147
- Poisson Boltzmann Beta, 147
- Poisson Ratio, 53, 60, 70, 210, 213
- Poisson ratio, 76
- Poisson-Boltzmann equation, 26, 144
- PoissonBEM, 293
- PoissonBEMSolver, 292
- PoissonBoltzmannSolve, 144
- Polyline Coordinates(n,DIM), 250, 254
- Polyline Divisions(n/2,DIM), 254
- Population Coefficient, 245
- Population Crossover, 245
- Population Size, 245
- Porosity Model, 290
- Porous Media, 30
- porous media, 25
- Porous Resistance, 30
- Post File, 261
- Potential, 106, 109, 147, 157, 294, 309
- Potential Coefficient, 28
- Potential Difference, 105, 143
- Potential Field, 28
- Potential Force, 28
- Potential Variable, 120
- Potential Variable Name, 202
- Power Control, 109
- Pre Strain, 54
- Pre Stress, 54
- Pressure, 28, 30, 76
- Pressure 1, 297
- Pressure 2, 297
- Pressure i, 30, 85, 300
- Pressure Source i, 85
- Principal Direction 2(3), 80
- Procedure, 34, 40, 45, 46, 52, 61, 70, 75, 81, 84, 92, 97, 100, 105, 109, 117, 119, 120, 124, 125, 129, 130, 134, 135, 137, 143, 146, 150, 156, 166, 172, 178–180, 184, 187, 190, 195, 200, 206, 210, 212, 215, 217, 222, 225, 228, 229, 231, 240, 241, 244, 248, 254, 256, 258, 262, 265, 268, 270, 272, 274, 277, 280, 283, 284, 286, 290, 291, 293, 296, 300, 304, 308, 311, 313, 320
- Project To Bottom, 285
- Projection Mask Variable, 285
- projection matrix, 233
- ProjectToPlane, 282

- Pseudo-Traction, 62
- Pull Rate Control, 191
- Quadratic Approximation, 117
- Radiation, 21
- Radiation Boundary, 21
- Radiation Boundary Open, 21
- Radiation External Temperature, 21
- Radiation Target Body, 21
- radiation: Linear System Keyword, 18
- Ratio of Convergence Tolerances, 94
- Rayleigh Damping, 53
- Rayleigh Damping Alpha, 54
- Rayleigh Damping Beta, 54
- Re Body Force i, 93
- Re Heat Source, 93
- Re Reference Wall Velocity i, 94
- Re Specific Acoustic Impedance, 93
- Re Specific Thermal Impedance, 94
- Re Surface Traction i, 93
- Re Temperature, 93
- Re Velocity i, 93
- reaction rate, 38
- reduced order model, 233
- Reference Pressure, 20, 29, 36, 46, 241
- Reference Temperature, 28, 54, 146
- Reference Wall Temperature, 94
- reinitialization, 176
- Reinitialize Field, 203
- Reinitialize Interval, 179
- Reinitialize Particles, 201
- Reinitialize Passive, 179
- Relative Permeability, 116, 133, 305
- Relative Permittivity, 105, 128, 143, 146, 161, 313
- Relative Permittivity im, 128
- Relative Reluctivity, 129
- Relative Reluctivity im, 129
- Relaxation Factor, 184, 245
- Reload Range Maximum, 270
- Reload Range Minimum, 270
- Reload Reading Intervals, 271
- Reload Solution File, 270
- Reload Starting Position, 270
- ReloadData, 270
- ReloadInput, 272
- ReloadSolution, 270
- Reluctivity, 117, 129, 133
- Reluctivity Im, 119
- Reluctivity im, 129
- Reset Interval i, 278
- Residual Reduction Ratio, 94
- Residual Water Content, 290
- Resitance, 124
- Restart File, 270
- Result Variable i, 207
- ResultOutputSolve, 258
- Reverse Ordering, 235
- Reynolds equation, 42
- Reynolds Pressure Variable Name, 46
- ReynoldsHeatingSolver, 42
- ReynoldsSolver, 42
- Richards equation, 288
- Richards Flux, 291
- Richards Source, 291
- RichardsFlux, 288
- RichardsSolver, 288
- Rigid Body, 234
- rigid body movement, 306
- RigidBodyReduction, 233
- RigidMeshMapper, 214
- Rotate Elasticity Tensor, 54
- Rotate Plane, 283
- run-time control, 272
- Runge Kutta, 207
- Saturated Hydraulic Conductivity, 290
- Saturated Initial Guess, 290
- Saturated Water Content, 290
- Save Axis, 254
- Save Boundaries Only, 260
- Save Bulk Only, 260
- Save Coil Index, 156
- Save Coil Set, 156
- Save component results, 251
- Save Coordinates(n,DIM), 249
- Save Displacements, 308
- Save Eigenfrequencies, 249
- Save Eigenvalues, 249
- Save Elemental Fields, 260
- Save Flux, 254
- Save Geometry Ids, 259
- Save Halo Elements Only, 260
- Save Isocurves, 254
- Save Line, 254
- Save Linear Elements, 260
- Save Mask, 254
- Save Nodal Fields, 260
- Save Points(n), 249
- Save Scalars, 251
- SaveBoundaryValues, 256
- SaveData, 247, 253, 256
- SaveGridData, 262
- SaveLine, 253
- SaveMaterials, 256
- SaveScalars, 247
- Scalar Field i, 204, 259, 263, 321
- Scalar Potential, 232
- ScalarPotentialSolver, 231
- Scalars File, 251

- Scalars Prefix, 248
- scalars: Keyword, 251
- Schur Complement Convergence Tolerance, 95
- secant method, 243
- Second Moment of Area 2, 80
- Second Moment of Area 3, 80
- Separate Loss Components, 150
- Set Constant Weight Sum, 280
- Shear Modulus, 80
- Shear Stress Output, 311
- Shear Stress Output File, 311
- Shell Thickness, 70
- ShellSolver, 63
- Show Norm Index, 249
- Show Variables, 259
- signed distance, 176
- Simplex Relative Length Scale, 245
- Simplex Restart Convergence Ratio, 245
- Simplex Restart Interval, 245
- Simpsons Rule, 150
- Simulation, 15, 34, 39, 60, 70, 80, 84, 92, 100, 125, 181, 222, 243, 251, 261, 296, 303
- Simulation Timestep Sizes, 202, 206
- Simulation Type, 15, 34, 39, 92, 243
- Sine Series i, 278
- Single Precision, 259, 320
- Skip Halo Elements, 260
- Slip Boundary, 94
- Slip Coefficient i, 168
- Smart Heater Control, 19
- Smart Heater Control After Tolerance, 17
- SmitcSolver, 72
- Solid, 191, 196
- Solver, 16, 26, 34, 39, 45, 46, 52, 61, 70, 75, 81, 84, 92, 94, 97, 100, 105, 109, 117, 119, 120, 124, 125, 129, 130, 133–135, 137, 146, 150, 155, 166, 172, 178–180, 184, 187, 190, 195, 200, 206, 210, 212, 215, 217, 222, 225, 227, 229, 231, 234, 240, 241, 244, 248, 253, 256, 258, 262, 265, 267, 268, 270, 272, 274, 277, 280, 283, 284, 286, 290, 291, 293, 296, 300, 304, 308, 311, 313, 315, 320
- SolveWithLinearRestriction, 314
- Sound Damping, 85, 97
- Sound Reaction Damping, 97
- Sound Source, 98
- Sound Speed, 85, 97
- Source, 294
- Source Acceleration, 98
- Source Gradient Correction, 207
- Source Name, 125
- Specific Heat, 93, 100
- Specific Heat Ratio, 20, 29, 36, 46, 93, 101
- Spring, 55, 76
- Stability Analysis, 52
- Stabilization Method, 184
- Stabilize, 17, 27, 35, 178
- Start Cycle i, 278
- Start Real Time, 278
- Start Real Time Fraction, 278
- Start Time i, 277
- Start Timestep i, 278
- StatCurrentSolve, 107
- StatCurrentSolver, 107
- StatElecBoundary, 140
- StatElecBoundaryCharge, 140
- StatElecBoundaryEnergy, 140
- StatElecBoundaryForce, 140
- StatElecBoundarySpring, 140
- StatElecForce, 312
- StatElecSolve, 103
- StatElecSolver, 103
- Static Magnetic Field, 305
- Statistical Info, 204
- StatMagSolver, 302
- Steady State Convergence Tolerance, 16, 27, 35, 40, 138, 240
- Stefan Boltzmann, 15
- Stefan-Boltzmann constant, 15
- Steinmetz loss model, 148
- Step Size, 245
- Stokes Stream Function, 223
- Stokes stream function, 221
- Stop Cycle i, 278
- Stop Time i, 278
- Stop Timestep i, 278
- Strain Load, 53
- Strain Reduction Operator, 71
- Stream Function First Node, 223
- Stream Function Penalty, 223
- Stream Function Scaling, 223
- Stream Function Shifting, 223
- Stream Function Velocity Variable, 223
- Streamline, 221
- StreamSolver, 221
- Stress Analysis, 53
- Stress Bodyforce 1, 53
- Stress Bodyforce 1 im, 53
- Stress Bodyforce 2, 53
- Stress Bodyforce 2 im, 53
- Stress Bodyforce 3, 53
- Stress Bodyforce 3 im, 53
- Stress Bodyforce j, 61
- Stress Load, 53, 55
- StressSolve, 49
- StressSolver, 49
- Structure Coupling With Boundary, 300

- Structure Interface, 86
- StructureMeshMapper, 217
- StructureProjectToPlane, 284
- substantial surface, 182
- Sum Forces, 311
- Surface Charge, 147
- Surface Charge Density, 106
- surface tension, 178
- Surface Tension Coefficient, 30
- Surface Tension Expansion Coefficient, 30
- Surface Traction i, 168
- Surface Traction k, 62
- Surface Velocity i, 45

- Table Format, 203, 263
- Tangent Velocity i, 45
- Target Field, 213
- Target Nodes, 232
- Target Variable, 135, 150, 225, 228, 229, 268, 280, 291
- Target Variable AV, 150
- Target Variable Direct, 150
- Target Variable i, 249, 275
- Temperature, 19, 20
- Temperature Lower Limit, 19
- Temperature Upper Limit, 19
- Tension, 76
- Tensor Field i, 259
- Theta i, 81
- Thickness, 76
- Time Derivative Order, 97
- Time Epsilon, 274
- Time Filter i, 278
- Time Multiplier, 274
- Time Name, 274
- Time Offset, 274
- Time Order, 206
- Time Point, 274
- Timestep Courant Number, 201
- Timestep Distance, 201
- Timestep Intervals, 244
- Timestep Size, 201, 206
- Timestepping Method, 16, 34, 39
- TimoshenkoSolver, 78
- Top Surface, 218
- Top Surface Level, 218
- Top Surface Variable Name, 218
- Torque Axis(3), 121
- Torque Origin(3), 121
- Torsional Constant, 80
- Transient Speedup, 191
- Translate Before Rotate, 215
- Transmissivity, 20, 21
- Triple Point Fixed, 191, 195

- U i, 71, 81
- UMAT Subroutine, 60
- Unit Charge, 146
- Update Gebhardt Factors, 17
- Update Transient System, 53
- Update View Factors, 17
- Use Density, 84
- Use Lagrange Gauge, 117
- Use Linear Elements, 184
- Use Nodal Loads, 191
- Use Original Coordinates, 215
- Use Piola Transform, 117, 129
- Use Tree Gauge, 117
- Use Velocity Laplacian, 167
- Use Wall Distance, 156
- User Defined Velocity, 138
- Utilize Previous Solution, 92

- van Genuchten Alpha, 290
- van Genuchten M, 290
- van Genuchten N, 290
- Variable, 34, 40, 45, 70, 75, 81, 84, 92, 97, 100, 105, 109, 117, 119, 129, 134, 137, 146, 150, 166, 172, 178, 180, 184, 187, 190, 195, 210, 212, 215, 222, 230, 231, 234, 241, 244, 257, 280, 290, 293, 296, 300, 304, 308
- Variable DOFs, 45, 70, 75, 81, 84, 97, 100, 105, 109, 137, 146, 167, 172, 184, 187, 190, 195, 222, 235, 244, 257, 290, 293, 296, 300, 308
- Variable Dofs, 92, 304
- Variable Global, 244
- Variable i, 207, 248, 254, 266, 275, 277, 285
- Variable_name, 40, 41
- Variable_name Gamma, 41
- Variable_name Lower Limit, 41
- Variable_name Source, 40
- Variable_name Upper Limit, 41
- variational inequality, 183
- Varname, 36, 185
- Varname Accumulation, 185
- Varname Accumulation Flux i, 185
- Varname Diffusion Source, 35
- Varname Diffusivity, 36
- Varname Flux, 36
- Varname Maximum Solubility, 36
- Varname Solubility Change Boundary, 37
- Varname Soret Diffusivity, 36
- Vector Field i, 204, 259, 263, 321
- VectorHelmholtz, 127
- VectorHelmholtzCalcFields, 127
- VectorHelmholtzSolver, 127
- Velocity 1, 161
- Velocity 2, 161

- Velocity 3, 161
- Velocity Assembly, 95
- Velocity Condition Variable Name, 202
- Velocity Convergence Tolerance, 95
- Velocity Field Name, 311
- Velocity Gradient Correction, 202, 207
- Velocity i, 28, 30
- Velocity Implicitness, 184
- Velocity Initialization Method, 201, 206
- Velocity Relaxation Factor, 191
- Velocity Smoothing Factor, 191
- Velocity Variable Name, 84, 202, 206, 284
- view factors, 17
- View Factors Fixed After Iterations, 18
- View Factors Fixed Tolerance, 18
- View Factors Geometry Tolerance, 18
- Viewfactor Combine Elements, 18
- Viewfactor Divide, 18
- Viscosity, 20, 28, 45, 93, 101, 161, 166, 172, 181, 311
- Viscosity Difference, 29, 181
- Viscosity Exponent, 29
- Viscosity Model, 29, 45, 166, 181
- Viscosity Temp Exp, 30
- Viscosity Temp Offset, 30
- Viscosity Temp Ref, 30
- Viscosity Transition, 29
- Volume Permutation, 283
- VorticitySolver, 227
- VTI, 262
- Vti Format, 262
- VTK, 258
- Vtk Format, 258
- VTU, 258, 262
- Vtu Format, 203, 258, 262
- Vtu Time Collection, 260
- vtu: Keyword, 261

- Wall Law, 31
- Wall Particle Bounciness, 203
- Wall Particle Collision, 204
- Wall Particle Radius, 203
- Wall Particle Spring, 203
- Water Content, 290
- Wave Flux 1,2, 85
- Wave Impedance 1,2, 85
- WaveFunctionSolver, 316
- WaveSolver, 96
- Weight Variable, 280
- WhitneyAVHarmonicSolver, 110
- WhitneyAVSolver, 110
- Wnodal, 300

- X Epsilon, 274
- X Name, 274

- XdmfWriter, 320
- Y Epsilon, 274
- Y Name, 274
- Yasuda Exponent, 29
- Youngs Modulus, 53, 60, 70, 80, 210, 213
- Youngs modulus, 76

- Z Epsilon, 274
- Z Name, 274
- Zeta Potential, 161