FEMLAB® 3

Multiphysics Modeling

ALE Models solved with FEMLAB 3

COMSOL

ALE Models

Introduction

To handle moving boundaries in some fluid flow problems you can apply the *arbitrary Lagrangian-Eulerian* (ALE) formulation. With this method in FEMLAB, field variables represent mesh displacement and mesh velocity. FEMLAB solves the equations on the original mesh but adjusts the equations to account for the mesh movements. The method covers situations in which the topology of the geometry does not change during the simulation and a suitable PDE for the mesh velocity exists.

This example considers two cases. In the first case, the mesh velocity is prescribed. In the second case, the mesh velocity varies freely.

Model Definition

Consider a mapping from the original domain Ω to the deformed domain $\overline{\Omega}$ defined by

$$x(X,Y,t),\ y(X,Y,t),\ X,\ Y \in \Omega,\ x,\ y \in \overline{\Omega},\ t \in R^{+}$$

with Jacobian inverse

$$\begin{bmatrix} I_{Xx} & I_{Xy} \\ I_{Yx} & I_{Yy} \end{bmatrix} = \frac{1}{D}\begin{bmatrix} y_Y & -x_Y \\ -y_X & x_X \end{bmatrix}, \text{ where } D = x_X\ y_Y - x_Y\ y_X .$$

The Navier-Stokes equations can be written as

$$\int_{\overline{\Omega}} (\tilde{u}_x(2\eta u_x - p) + \tilde{v}_y(2\eta v_y - p) + (\hat{v}_x + \hat{u}_y)\eta(u_y + v_x) +$$
$$\hat{u}((u - \psi_x)u_x + (v - \psi_y)u_y) + \hat{v}((u - \psi_x)v_x + (v - \psi_y)v_y) +$$
$$-\hat{p}(u_x + v_y) + \rho\hat{u}\frac{\partial u}{\partial t} + \rho\hat{v}\frac{\partial v}{\partial t} - \hat{u}f_x - \hat{v}f_y)d\Omega = 0$$

in the moving coordinate system, where $\psi = (\psi_x, \psi_y)$ is the mesh velocity [2] and $\hat{}$ denotes a test function. The deformed velocity derivatives can be expressed using the original velocity derivatives

$$\begin{bmatrix} u_x & u_y \\ v_x & v_y \end{bmatrix} = \begin{bmatrix} u_X I_{Xx} + u_Y I_{Yx} & u_X I_{Xy} + u_Y I_{Yy} \\ v_X I_{Xx} + v_Y I_{Yx} & v_X I_{Xy} + v_Y I_{Yy} \end{bmatrix} .$$

The integration of the weak equations on the original domain

$$\begin{aligned} &\int_\Omega D((\hat{u}_X I_{Xx} + \hat{u}_Y I_{Yx})(2\eta u_x - p) + (\hat{v}_X I_{Xy} + \hat{v}_Y I_{Yy})(2\eta v_y - p) + \\ &(\hat{u}_X I_{Xy} + \hat{u}_Y I_{Yy} + \hat{v}_X I_{Xx} + \hat{v}_Y I_{Yx})\eta(u_y + v_x) + \\ &\hat{u}((u - \psi_x)u_x + (v - \psi_y)u_y) + \hat{v}((u - \psi_x)v_x + (v - \psi_y)v_y) + \\ &-\hat{p}(u_x + v_y) + \rho\hat{u}\frac{\partial u}{\partial t} + \rho\hat{v}\frac{\partial v}{\partial t} - \hat{u}f_x - \hat{v}f_y)d\Omega = 0 \end{aligned}$$

derives a set of equations that can be solved on the fixed mesh. Note that referring to $\hat{u}_x$ here would seem logical but it introduces extra terms that complicate the equations for the mesh discussed below.

Prescribed Moving Boundaries

To model a prescribed moving boundary and solve for the mesh velocity using a Poisson's equation in the moving coordinate system, use the following equations:

$$\begin{aligned} &\int_\Omega D((\hat{\psi}_{xX} I_{Xx} + \hat{\psi}_{xY} I_{Yx})\psi_{xx} + (\hat{\psi}_{xX} I_{Xy} + \hat{\psi}_{xY} I_{Yy})\psi_{xy})d\Omega = 0 \\ &\int_\Omega D((\hat{\psi}_{yX} I_{Xx} + \hat{\psi}_{yY} I_{Yx})\psi_{yx} + (\hat{\psi}_{yX} I_{Xy} + \hat{\psi}_{yY} I_{Yy})\psi_{yy})d\Omega = 0 \qquad \text{on } \Omega \\ &\frac{\partial x}{\partial t} = \psi_x \\ &\frac{\partial y}{\partial t} = \psi_y \end{aligned}$$

with initial condition $x = X, y = Y$. Solving for the mesh movements in the moving coordinate systems makes the problem nonlinear, but often allows for larger mesh movements. Also provide a velocity field for the boundary

$$
\left.\begin{aligned}
\psi_x &= U(t) \\
\psi_y &= V(t) \\
u &= \psi_x \\
v &= \psi_y
\end{aligned}\right\} \quad \text{on } \partial\overline{\Omega}\,.
$$

The pressure must be fixed at one point for the problem to be well-defined because there is no force on the boundary.

Free Moving Boundaries

To model a free moving boundary and solve for the mesh velocity using a Poisson's equation in the fixed coordinate system, refer to the equations

$$
\boxed{\begin{aligned}
&\int_\Omega (\hat{\psi}_{xX}\psi_{xX} + \hat{\psi}_{xY}\psi_{xY})\mathrm{d}\Omega = 0 \\
&\int_\Omega (\hat{\psi}_{yX}\psi_{yX} + \hat{\psi}_{yY}\psi_{yY})\mathrm{d}\Omega = 0 \qquad \text{on } \Omega \\
&\qquad\qquad \frac{\partial x}{\partial t} = \psi_x \\
&\qquad\qquad \frac{\partial y}{\partial t} = \psi_y
\end{aligned}}
$$

with initial condition $x = X$, $y = Y$. Solving for the mesh movement in the fixed coordinate system simplifies the equations for the mesh but allows for smaller mesh movements.

For the boundary, model the simplest possible case (no wall, no surface tension) of free boundaries by adding additional boundary conditions and an equation for the velocities of the free boundary.

$$
\boxed{\begin{aligned}
\psi_x &= u \\
\psi_y &= v
\end{aligned}}
$$

These constraints are implemented in weak form using Lagrange multipliers l_x and l_y. The corresponding weak terms $\Psi_x l_x$ and $\Psi_y l_y$ are added to the weak form of the PDE for the mesh movement. This means that the flow moves freely at the free boundary and takes care of ensuring that the constraint only affects the mesh movement. The reason to use non-ideal weak constraints here is to avoid constraint forces in the Navier-Stokes equations.

Modeling in FEMLAB

PRESCRIBED MOVING BOUNDARIES

Consider a circular domain (radius 1) with a moving smaller circular domain (radius 0.2) subtracted. The smaller circular domain is moving according to the following pattern:

$$U = A\frac{2}{\pi}\left(-\operatorname{atan}(t)\sin(t) + \frac{1}{1+t^2}\cos(t)\right)$$
$$V = A\frac{2}{\pi}\left(\operatorname{atan}(t)\cos(t) + \frac{1}{1+t^2}\sin(t)\right)$$

Because the mesh movement does not affect the fluid flow, you can solve for the mesh movement first and then for the fluid field. The constant $A = 0.2$. The formulas come from

$$x = r(t)\cos(t)$$
$$y = r(t)\sin(t)$$

where $r(t) = A\frac{2}{\pi}\operatorname{atan}(t)$. The force affecting the fluid is $f_x = 0, f_y = 0$.

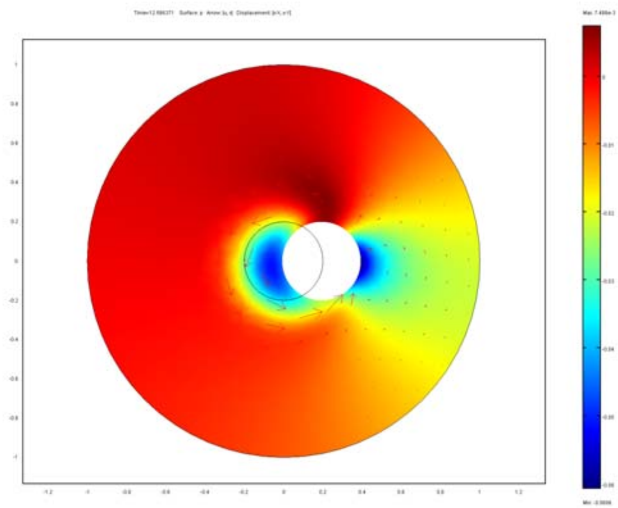
FREE MOVING BOUNDARIES

Model a circular domain (radius 0.7) with a fluid about it. A “gravity-like force” keeps the fluid on top of the circular domain ($f_x = -10x, f_y = -10y$). The motion of the fluid is triggered by an initial velocity.

Results

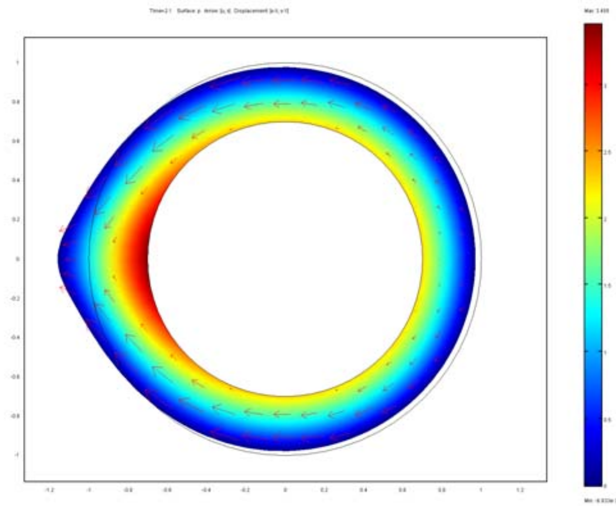
PRESCRIBED MOVING BOUNDARIES

The interior domain is moving outward in a spiral-like motion, approaching a circular movement. You can verify the steady-state solution by solving a stationary Navier-Stokes problem in a rotating coordinate system.



FREE MOVING BOUNDARIES

The plot shows the free fluid surface after 2.1 time units. Notice that the pressure appears to depend mostly on the depth of the fluid. This simplified model avoids the need to represent the interaction between the fluid and walls. While often you also need to include a surface tension model, this example runs without defining surface tension impacts.



References

[1] J. Donea, A. Huerta, J.-Ph. Ponthot, A. Rodrígues-Ferran, "Arbitrary Langrangian-Eulerian Methods", *Encyclopedia of Computational Mechanics*, Edited by Erwin Stein, René de Borst and Thomas J.R. Hughes, John Wiley & Sons, 2004.

[2] M.A. Fernández, M. Moubachir, "Sensitivity Analysis for an Incompressible Aeroelastic Systems", *Mathematical Models and Methods in Applied Sciences*, 12, no 8, 2002.

Model Library path: `FEMLAB/Fluid_Dynamics/ale_prescribed`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator**, select **2D** in the **Space dimension** list.

2 In the **PDE Modes** folder, select **Weak Form, Subdomain**.

3 Type `psix psiy x y` in the **Dependent variables** edit field.

4 Type `mesh` in the **Application mode name** edit field.

5 Click the **Multiphysics** button and then the **Add Geometry** button.

6 Keep the 2D space dimension and enter `X Y Z` as independent variables.

7 Click **OK**.

8 Click **Add**.

9 In the **PDE Modes** folder, select **Weak Form, Subdomain**.

10 Type `u v p` in the **Dependent variables** edit field.

11 Type `ns` in the **Application mode name** edit field.

12 Click **Add**.

13 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Constants**.

2 In the **Constants** dialog box, define the following constants with names and expressions:

NAME	EXPRESSION
nu	0.001
A	0.2

3 Click **OK**.

GEOMETRY MODELING

1 Shift-click on the **Ellipse/Circle (Centered)** button in the Draw toolbar. Enter radius 1, and keep the default origin.

2 Shift-click on the **Ellipse/Circle (Centered)** button in the Draw toolbar. Enter radius 0.2, and keep the default origin.

3 Select both domains by pressing **Ctrl+A** and then click the **Difference** button.

PHYSICS SETTINGS

Expression Variables

1 On the **Options** menu, point to **Expressions** and then click **Subdomain Expressions**.

2 Enter the following expression variables for subdomain 1:

SUBDOMAIN	1
dJ	xX*yY-xY*yX
iJXx	yY/dJ
iJXy	-xY/dJ

SUBDOMAIN	1
iJYx	-yX/dJ
iJYy	xX/dJ
ux	uX*iJXx+uY*iJYx
uy	uX*iJXy+uY*iJYy
vx	vX*iJXx+vY*iJYx
vy	vX*iJXy+vY*iJYy
U	A/(pi/2)*(-atan(t)*sin(t)+1/(1+t^2)*cos(t))
V	A/(pi/2)*(atan(t)*cos(t)+1/(1+t^2)*sin(t))
psixx	psixX*iJXx+psixY*iJYx
psixy	psixX*iJXy+psixY*iJYy
psiyx	psiyX*iJXx+psiyY*iJYx
psiyy	psiyX*iJXy+psiyY*iJYy

3 Click **OK**.

Subdomain Settings

1 From the **Multiphysics** menu, choose **Weak Form, Subdomain (mesh)**.

2 In the **Subdomain Settings** dialog box, enter the following settings:

SUBDOMAIN	1
weak(1)	dJ*((test(psixX)*iJXx+test(psixY)*iJYx)*psixx+(test(psixX)*iJXy+test(psixY)*iJYy)*psixy)
weak(2)	dJ*((test(psiyX)*iJXx+test(psiyY)*iJYx)*psiyx+(test(psiyX)*iJXy+test(psiyY)*iJYy)*psiyy)
weak(3)	test(x)*psix
weak(4)	test(y)*psiy
dweak(1)	0
dweak(2)	0
dweak(3)	test(x)*x_time
dweak(4)	test(y)*y_time
init(1)	0
init(2)	0
init(3)	X
init(4)	Y

3 Click **OK**.

4 From the **Multiphysics** menu, choose **Weak Form, Subdomain (ns)**.

5 In the **Subdomain Settings** dialog box, enter the following settings:

SUBDOMAIN	1
weak(1)	`-dJ*((test(uX)*iJXx+test(uY)*iJYx)*(2*nu*ux-p)+(test (uX)*iJXy+test(uY)*iJYy)*nu*(uy+vx)+test(u)*((u-psix )*ux+(v-psiy)*uy))`
weak(2)	`-dJ*((test(vX)*iJXy+test(vY)*iJYy)*(2*nu*vy-p)+(test (vX)*iJXx+test(vY)*iJYx)*nu*(uy+vx)+test(v)*((u-psix )*vx+(v-psiy)*vy))`
weak(3)	`-dJ*(-test(p)*(ux+vy))`
dweak(1)	`u_time*u_test*dJ`
dweak(2)	`v_time*v_test*dJ`
dweak(3)	`0`
shape	`shlag(2,'u') shlag(2,'v') shlag(2,'p')`
gporder	`4 4 2`
cporder	`2 2 1`

6 Click **OK**.

Boundary Conditions

1 From the **Multiphysics** menu, choose **Weak Form, Subdomain (mesh)**.

2 In the **Boundary Settings** dialog box, enter the following settings:

BOUNDARY	1,2,5,8	3,4,6,7
constr(1)	`psix`	`psix-U`
constr(2)	`psiy`	`psiy-V`
constr(3)	`0`	`0`
constr(4)	`0`	`0`

3 Click **OK**.

4 From the **Multiphysics** menu, choose **Weak Form, Subdomain (ns)**.

5 In the **Boundary Settings** dialog box, enter the following settings:

BOUNDARY	1,2,5,8	3,4,6,7
constr(1)	`u`	`u-U`
constr(2)	`v`	`v-V`
constr(3)	`0`	`0`

6 Click **OK**.

Point Settings

1 Open the **Point Settings** dialog box.

2 Select point 1, click the **Constr** tab, and type `p` in the third (bottom) edit field.

3 Click **OK**.

MESH GENERATION

1 Open the **Mesh Parameters** dialog box.

2 Type 0.1 in the **Mesh curvature factor** edit field.

3 Click **OK**.

4 Click the **Initialize Mesh** toolbar button.

COMPUTING THE SOLUTION

Start by computing the mesh displacements and mesh velocity, and then use it during the actual fluid flow computation.

1 From the **Solve** menu, choose **Solver Parameters**.

2 In the **Solver** list, select **Time dependent**.

3 Enter `0 2*2*pi` in the **Times** edit field.

4 Enter `1e-4` in both the **Relative tolerance** and **Absolute tolerance** edit fields.

5 Click the **Time Stepping** tab.

6 Select **Time steps from solver** in the **Times to store in output** list.

7 Type `3` in the **Maximum BDF order** edit field.

8 Select **Yes** in the **Singular mass matrix** list.

9 Select **Backward Euler** in the **Consistent initialization of DAE systems** list.

10 Select **Exclude Algebraic** in the **Error estimation strategy** list.

11 Click the **Advanced** tab.

12 In the **Scaling of variables** area, select **None**.

13 Click **OK**.

14 From the **Solve** menu, choose **Solver Manager**.

15 Click the **Initial value expression** option button in the **Initial value** area and the **Use setting from Initial value frame** option button in the **Value of variables not solved for and linearization point** area.

16 Click the **Solve For** tab.

17 Select **Weak form, Subdomain (mesh)**.

18 Click **OK**.

19 Click the **Solve** button.

20 From the **Solve** menu, choose **Solver Parameters**.

21 Click the **General** tab.

22 Type `linspace(0,2*2*pi,101)` in the **Times** edit field.

23 Click the **Time Stepping** tab.

24 Enter `5` in the **Maximum BDF order** edit field.

25 Select **Specified times** in the **Times to store in output** list.

26 Click **OK**.

27 From the **Solve** menu, choose **Solver Manager**.

28 Click the **Current solution** option button in the **Initial value** frame.

29 in the **Values of variables not solved for and linearization point** area, click the **Current solution** option button and select **All** in the **Solution at time** list.

30 Click the **Solve For** tab.

31 Select **Weak form, Subdomain (ns)**.

32 Click **OK**.

33 Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

1 Open the **Plot Parameters** dialog box.

2 On the **Surface** tab, enter the expression `p` in **Surface data**, **Expression**.

3 On the **Arrow** tab, enter `u` and *v* on the **Subdomain** tab. Check **Arrow plot**.

4 On the **Deform** tab, enter `x-X` and `y-Y` on the **Subdomain** tab. Check **Subdomain** and uncheck **Boundary**. Check **Deformed shape plot**.

5 Click **OK**.

6 Click the **Animation** toolbar button.

Model Library path: `FEMLAB/Fluid_Dynamics/ale_free_boundary`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator**, select **2D** in the **Space dimension** list.

2 In the **PDE Modes** folder, select **Weak Form, Subdomain**.

3 Type `psix psiy x y` in the **Dependent variables** edit field.

4 Click the **Multiphysics** button and then the **Add Geometry** button.

5 Keep the 2D space dimension, and enter `X Y Z` as independent variables.

6 Click **OK**.

7 Type `mesh` in the **Application mode name** edit field.

8 Click **Add**.

9 In the **PDE Modes** folder, select **Weak Form, Subdomain**.

10 Type `u v p` in the **Dependent variables** edit field.

11 Type **ns** in the **Application mode name** edit field.

12 Click **Add**.

13 In the **PDE Modes** folder, select **Weak Form, Boundary**.

14 Type `lx ly` in the **Dependent variables** edit field.

15 Type **free** in the **Application mode name** edit field.

16 Click **Add**.

17 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Constants**.

2 In the **Constants** dialog box, define the following constants with names and expressions:

NAME	EXPRESSION
nu	0.01
A	1

3 Click **OK**.

GEOMETRY MODELING

1 Shift-click on the **Ellipse/Circle (Centered)** button in the Draw toolbar. Enter radius 1, and keep the default origin.

2 Shift-click on the **Ellipse/Circle (Centered)** button in the Draw toolbar. Enter radius 0.7, and keep the default origin.

3 Press **Ctrl+A** to select both domains and then click the **Difference** button.

PHYSICS SETTINGS

Expression Variables

1 On the **Options** menu, point to **Expressions** and then click **Subdomain Expressions**.

2 Enter the following expression variables for subdomain 1:

SUBDOMAIN	1
dJ	xX*yY-xY*yX
iJXx	yY/dJ
iJXy	-xY/dJ
iJYx	-yX/dJ
iJYy	xX/dJ
ux	uX*iJXx+uY*iJYx
uy	uX*iJXy+uY*iJYy
vx	vX*iJXx+vY*iJYx
vy	vX*iJXy+vY*iJYy

3 Click **OK**.

Subdomain Settings

1 From the **Multiphysics** menu, choose **Weak form, Subdomain (mesh)**.

2 In the **Subdomain Settings** dialog box, enter the following settings:

SUBDOMAIN	1
weak(1)	test(psixX)*psixX+test(psixY)*psixY
weak(2)	test(psiyX)*psiyX+test(psiyY)*psiyY
weak(3)	test(x)*psix
weak(4)	test(y)*psiy
dweak(1)	0
dweak(2)	0
dweak(3)	test(x)*x_time
dweak(4)	test(y)*y_time
init(1)	0
init(2)	0
init(3)	X
init(4)	Y

3 Click **OK**.

4 From the **Multiphysics** menu, choose **Weak Form, Subdomain (ns)**.

5 In the **Subdomain Settings** dialog box, enter the following settings:

SUBDOMAIN	1
weak(1)	`-dJ*((test(uX)*iJXx+test(uY)*iJYx)*(2*nu*ux-p)+(test (uX)*iJXy+test(uY)*iJYy)*nu*(uy+vx)+test(u)*((u-psi) *ux+(v-psiy)*uy+10*x))`
weak(2)	`-dJ*((test(vX)*iJXy+test(vY)*iJYy)*(2*nu*vy-p)+(test (vX)*iJXx+test(vY)*iJYx)*nu*(uy+vx)+test(v)*((u-psi) *vx+(v-psiy)*vy+10*y))`
weak(3)	`-dJ*(-test(p)*(ux+vy))`
dweak(1)	`u_time*u_test*dJ`
dweak(2)	`v_time*v_test*dJ`
dweak(3)	`0`
init(1)	`A`
init(2)	`0`
init(3)	`0`
shape	`shlag(2,'u') shlag(2,'v') shlag(1,'p')`
gporder	`4 4 2`
cporder	`2 2 1`

6 Click **OK**.

Boundary Conditions

1 From the **Multiphysics** menu, choose **Weak Form, Subdomain (mesh)**.

2 In the **Boundary Settings** dialog box, enter the following settings:

BOUNDARY	1,2,5,8	3,4,6,7
constr(1)	`0`	`psix`
constr(2)	`0`	`psiy`
constr(3)	`0`	`0`
constr(4)	`0`	`0`
weak(1)	`test(psix)*lx`	`0`
weak(2)	`test(psiy)*ly`	`0`
weak(3)	`0`	`0`
weak(4)	`0`	`0`

3 Click **OK**.

4 From the **Multiphysics** menu, choose **Weak Form, Subdomain (ns)**.

5 In the **Boundary Settings** dialog box, enter the following settings:

BOUNDARY	1,2,5,8	3,4,6,7
constr(1)	0	u
constr(2)	0	v
constr(3)	0	0

6 Click OK.

7 From the **Multiphysics** menu, choose **Weak form, Boundary (free)**.

8 In the **Boundary Settings** dialog box, enter the following settings

BOUNDARY	1,2,5,8	3,4,6,7
weak(1)	`test(lx)*(psix-u)`	0
weak(2)	`test(ly)*(psiy-v)`	0

9 Clear the **Active in this domain** check box for boundaries 3, 4, 6, and 7.

10 Click **OK**.

MESH GENERATION

1 Open the Mesh Parameters dialog box.

2 Set the **Maximum element size** to 0.1.

3 Click **OK**.

4 Click the **Initialize Mesh** toolbar button.

COMPUTING THE SOLUTION

1 From the **Solve** menu, choose **Solver Parameters**.

2 Select **Time dependent** from the **Solver** list.

3 Type `linspace(0,10,101)` in the **Times** edit field.

4 Type `1e-6` in both the **Relative tolerance** and the **Absolute tolerance** edit field.

5 Click the **Time Stepping** tab.

6 Select **Yes** in the **Singular mass matrix** list.

7 Select **Backward Euler** in the **Consistent initialization of DAE systems** list.

8 Select **Exclude algebraic** in the **Error estimation strategy** list.

9 Click the **Advanced** tab.

10 In the **Scaling of variables** area, select **None**.

11 Click **OK**.

12 Click the **Solve** button.

POSTPROCESSING AND VISUALIZATION

1 Open the **Plot Parameters** dialog box.

2 On the **Surface** tab, type the expression `p` in the **Expression** edit field in the **Surface data** area.

3 On the **Arrow** tab, enter `u` and `v` on the **Subdomain** tab. Select the **Arrow plot** check box.

4 On the **Deform** tab, type `x-X` and `y-Y` on the **Subdomain** tab. Select the **Subdomain** check box and clear the **Boundary** check box. Select the **Deformed shape plot** check box.

5 Click **OK**.

Click the **Animation** toolbar button.