Chapter 1

Demo problem: Boundary-driven elastic deformation of a fish-shaped domain

Detailed documentation to be written. Here's a plot of the result and the already fairly well documented driver code...

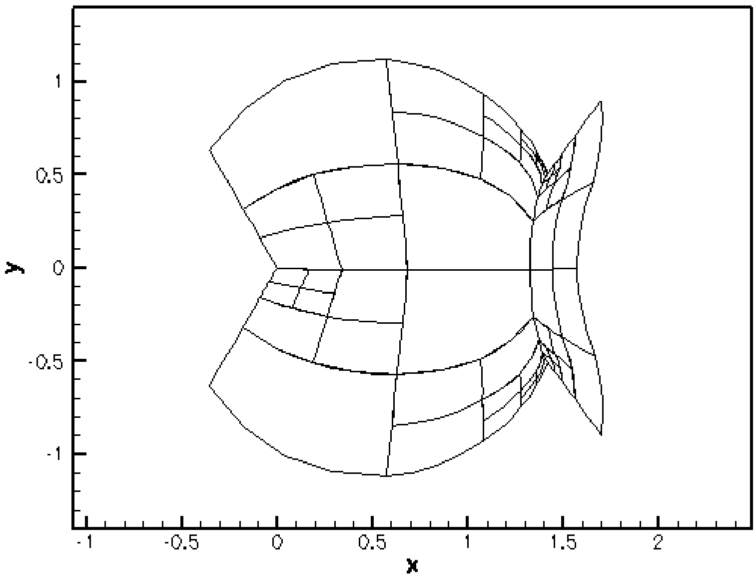

Figure 1.1 Boundary-driven elastic deformation of a fish-shaped domain.

```
//LIC// ====================================================================
//LIC// This file forms part of oomph-lib, the object-oriented,
//LIC// multi-physics finite-element library, available
//LIC// at http://www.oomph-lib.org.
//LIC//
//LIC// Copyright (C) 2006-2022 Matthias Heil and Andrew Hazel
//LIC//
//LIC// This library is free software; you can redistribute it and/or
//LIC// modify it under the terms of the GNU Lesser General Public
//LIC// License as published by the Free Software Foundation; either
//LIC// version 2.1 of the License, or (at your option) any later version.
//LIC//
//LIC// This library is distributed in the hope that it will be useful,
//LIC// but WITHOUT ANY WARRANTY; without even the implied warranty of
//LIC// MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.  See the GNU
//LIC// Lesser General Public License for more details.
//LIC//
//LIC// You should have received a copy of the GNU Lesser General Public
```

```
//LIC// License along with this library; if not, write to the Free Software
//LIC// Foundation, Inc., 51 Franklin Street, Fifth Floor, Boston, MA
//LIC// 02110-1301  USA.
//LIC//
//LIC// The authors may be contacted at oomph-lib@maths.man.ac.uk.
//LIC//
//LIC//====================================================================
// Driver for elastic deformation of a fish-shaped domain with
// adaptivity -- deformation is driven by displacement of
// GeomObject-based boundary!

// Generic oomph-lib headers
#include "generic.h"

// Solid mechanics
#include "solid.h"

// The fish mesh
#include "meshes/fish_mesh.h"

using namespace std;

using namespace oomph;


/// ////////////////////////////////////////////////////////////////////
/// ////////////////////////////////////////////////////////////////////
/// ////////////////////////////////////////////////////////////////////

//======================================================================
/// Refineable fish mesh upgraded to become a solid mesh
//======================================================================
template<class ELEMENT>
class ElasticFishMesh : public virtual RefineableFishMesh<ELEMENT>,
                        public virtual SolidMesh
{

public:

 /// Constructor: Build underlying adaptive fish mesh and then
 /// set current Eulerian coordinates to be the Lagrangian ones.
 /// Pass pointer to geometric objects that specify the
 /// fish's back in the "current" and "undeformed" configurations,
 /// and pointer to timestepper (defaults to Static)
 // Note: FishMesh is virtual base and its constructor is automatically
 // called first! --> this is where we need to build the mesh;
 // the constructors of the derived meshes don't call the
 // base constructor again and simply add the extra functionality.
 ElasticFishMesh(GeomObject* back_pt, GeomObject* undeformed_back_pt,
                 TimeStepper* time_stepper_pt=&Mesh::Default_TimeStepper) :
  FishMesh<ELEMENT>(back_pt,time_stepper_pt),
  RefineableFishMesh<ELEMENT>(back_pt,time_stepper_pt)
  {
   // Mesh has been built, adaptivity etc has been set up -->
   // assign the Lagrangian coordinates so that the current
   // configuration becomes the stress-free initial configuration
   set_lagrangian_nodal_coordinates();

   // Build "undeformed" domain: This is a "deep" copy of the
   // Domain that we used to create set the Eulerian coordinates
   // in the initial mesh -- the original domain (accessible via
   // the private member data Domain_pt) will be used to update
   // the position of the boundary nodes; the copy that we're
   // creating here will be used to determine the Lagrangian coordinates
   // of any newly created SolidNodes during mesh refinement
   double xi_nose = this->Domain_pt->xi_nose();
   double xi_tail = this->Domain_pt->xi_tail();
   Undeformed_domain_pt=new FishDomain(undeformed_back_pt,xi_nose,xi_tail);

   // Loop over all elements and set the undeformed macro element pointer
   unsigned n_element=this->nelement();
   for (unsigned e=0;e<n_element;e++)
    {
     // Get pointer to full element type
     ELEMENT* el_pt=dynamic_cast<ELEMENT*>(this->element_pt(e));

     // Set pointer to macro element so the curvlinear boundaries
     // of the undeformed mesh/domain get picked up during adaptive
     // mesh refinement
     el_pt->set_undeformed_macro_elem_pt(
      Undeformed_domain_pt->macro_element_pt(e));

     // Use MacroElement representation for
     // Lagrangian coordinates of newly created
     // nodes
     el_pt->enable_use_of_undeformed_macro_element_for_new_lagrangian_coords();
    }
```

```
  }


 /// Destructor: Kill "undeformed" Domain
 virtual ~ElasticFishMesh()
  {
   delete Undeformed_domain_pt;
  }


private:

 /// Pointer to "undeformed" Domain -- used to determine the
 /// Lagrangian coordinates of any newly created SolidNodes during
 /// Mesh refinement
 Domain* Undeformed_domain_pt;

};




/// ////////////////////////////////////////////////////////////////////
/// ////////////////////////////////////////////////////////////////////
/// ////////////////////////////////////////////////////////////////////




//================================================================
/// Global variables
//================================================================
namespace Global_Physical_Variables
{
 /// Pointer to strain energy function
 StrainEnergyFunction* Strain_energy_function_pt;

 /// Pointer to constitutive law
 ConstitutiveLaw* Constitutive_law_pt;

 /// Elastic modulus
 double E=1.0;

 /// Poisson's ratio
 double Nu=0.3;

 /// "Mooney Rivlin" coefficient for generalised Mooney Rivlin law
 double C1=1.3;

 /// Body force
 double Gravity=0.0;

 /// Body force vector: Vertically downwards with magnitude Gravity
 void body_force(const double& t,
                 const Vector<double>& xi,
                 Vector<double>& b)
 {
  b[0]=0.0;
  b[1]=-Gravity;
 }

}


/// ////////////////////////////////////////////////////////////////////
/// ////////////////////////////////////////////////////////////////////
/// ////////////////////////////////////////////////////////////////////



//======================================================================
/// Boundary-driven elastic deformation of fish-shaped domain.
//======================================================================
template<class ELEMENT>
class ElasticFishProblem : public Problem
{

public:

 /// Constructor:
 ElasticFishProblem();

 /// Run simulation.
 void run();

 /// Access function for the mesh
 ElasticFishMesh<ELEMENT>* mesh_pt()
```

```
  {return dynamic_cast<ElasticFishMesh<ELEMENT>*>(Problem::mesh_pt());}

 /// Doc the solution
 void doc_solution(DocInfo& doc_info);

 /// Update function (empty)
 void actions_after_newton_solve() {}

 /// Update before solve: We're dealing with a static problem so
 /// the nodal positions before the next solve merely serve as
 /// initial conditions. For meshes that are very strongly refined
 /// near the boundary, the update of the displacement boundary
 /// conditions (which only moves the SolidNodes *on* the boundary),
 /// can lead to strongly distorted meshes. This can cause the
 /// Newton method to fail --> the overall method is actually more robust
 /// if we use the nodal positions as determined by the Domain/MacroElement-
 /// based mesh update as initial guesses.
 void actions_before_newton_solve()
  {
   bool update_all_solid_nodes=true;
   mesh_pt()->node_update(update_all_solid_nodes);
  }

 /// Update after adapt: Pin all redundant pressure nodes (if required)
 void actions_after_adapt()
  {
   // Pin the redundant solid pressures (if any)
   PVDEquationsBase<2>::pin_redundant_nodal_solid_pressures(
    mesh_pt()->element_pt());
  }


private:

 // Geometric object that represents the deformable fish back
 Circle* Fish_back_pt;

};

//======================================================================
/// Constructor:
//======================================================================
template<class ELEMENT>
ElasticFishProblem<ELEMENT>::ElasticFishProblem()
{

 // Set coordinates and radius for the circle that will become the fish back
 double x_c=0.5;
 double y_c=0.0;
 double r_back=1.0;

 // Build geometric object that will become the deformable fish back
 //GeomObject* fish_back_pt=new ElasticFishBackElement(x_c,y_c,r_back);
 Fish_back_pt=new Circle(x_c,y_c,r_back);

 // Build geometric object that specifies the fish back in the
 // undeformed configuration (basically a deep copy of the previous one)
 GeomObject* undeformed_fish_back_pt=new Circle(x_c,y_c,r_back);

 // Build fish mesh with geometric objects that specify the deformable
 // and undeformed fish back
 Problem::mesh_pt()=new ElasticFishMesh<ELEMENT>(Fish_back_pt,
                                                  undeformed_fish_back_pt);

 // Set error estimator
 Z2ErrorEstimator* error_estimator_pt=new Z2ErrorEstimator;
 mesh_pt()->spatial_error_estimator_pt()=error_estimator_pt;

 // Change/doc targets for mesh adaptation
 mesh_pt()->max_permitted_error()=0.05;
 mesh_pt()->min_permitted_error()=0.005;
 mesh_pt()->doc_adaptivity_targets(cout);

 // Pin all nodal positions apart from those on the tail
 unsigned num_bound = mesh_pt()->nboundary();
 for(unsigned ibound=0;ibound<num_bound;ibound++)
  {
   if (ibound!=2)
    {
     unsigned num_nod=mesh_pt()->nboundary_node(ibound);
     for (unsigned inod=0;inod<num_nod;inod++)
      {
       for (unsigned i=0;i<2;i++)
        {
         mesh_pt()->boundary_node_pt(ibound,inod)->pin_position(i);
        }
      }
```

```
    }
  }

 //Loop over the elements in the mesh to set parameters/function pointers
 unsigned  n_element =mesh_pt()->nelement();
 for(unsigned i=0;i<n_element;i++)
  {
   //Cast to a solid element
   ELEMENT *el_pt = dynamic_cast<ELEMENT*>(mesh_pt()->element_pt(i));

   // Set the constitutive law
   el_pt->constitutive_law_pt() =
    Global_Physical_Variables::Constitutive_law_pt;

   // Set the body force
   el_pt->body_force_fct_pt()=Global_Physical_Variables::body_force;

  }

 // Pin the redundant solid pressures (if any)
 PVDEquationsBase<2>::pin_redundant_nodal_solid_pressures(
  mesh_pt()->element_pt());

 //Attach the boundary conditions to the mesh
 cout « assign_eqn_numbers() « std::endl;

 // Refine the problem uniformly (this automatically passes the
 // function pointers/parameters to the finer elements
 refine_uniformly();

 // The non-pinned positions of the newly SolidNodes will have been
 // determined by interpolation. Update all solid nodes based on
 // the Mesh's Domain/MacroElement representation.
 bool update_all_solid_nodes=true;
 mesh_pt()->node_update(update_all_solid_nodes);

 // Now set the Eulerian equal to the Lagrangian coordinates
 mesh_pt()->set_lagrangian_nodal_coordinates();
}


//==================================================================
/// Doc the solution
//==================================================================
template<class ELEMENT>
void ElasticFishProblem<ELEMENT>::doc_solution(DocInfo& doc_info)
{

 ofstream some_file;
 char filename[100];

 // Number of plot points
 unsigned npts = 5;

 // Output shape of deformed body
 sprintf(filename,"%s/soln%i.dat",doc_info.directory().c_str(),
         doc_info.number());
 some_file.open(filename);
 mesh_pt()->output(some_file,npts);
 some_file.close();

 // removed until Jacobi eigensolver is re-instated
 // // Output principal stress vectors at the centre of all elements
 // SolidHelpers::doc_2D_principal_stress<ELEMENT>(doc_info,mesh_pt());

}

//==================================================================
/// Run the problem
//==================================================================
template<class ELEMENT>
void ElasticFishProblem<ELEMENT>::run()
{

 // Output
 DocInfo doc_info;

 // Set output directory
 doc_info.set_directory("RESLT");

 // Step number
 doc_info.number()=0;
 // Initial parameter values
 // Gravity:
 Global_Physical_Variables::Gravity=0.1;
 //Parameter incrementation
 unsigned nstep=5;
```

```
 for(unsigned i=0;i<nstep;i++)
  {
   //Solve the problem with Newton's method, allowing for up to 5
   //rounds of adaptation
   newton_solve(5);

   // Doc solution
   doc_solution(doc_info);
   doc_info.number()++;

   // Increment width
   Fish_back_pt->y_c()+=0.03;
  }

}

//======================================================================
/// Driver for simple elastic problem
//======================================================================
int main()
{

 //Initialise physical parameters
 Global_Physical_Variables::E  = 2.1;
 Global_Physical_Variables::Nu = 0.4;
 Global_Physical_Variables::C1 = 1.3;
 // Define a strain energy function: Generalised Mooney Rivlin
 Global_Physical_Variables::Strain_energy_function_pt =
  new GeneralisedMooneyRivlin(&Global_Physical_Variables::Nu,
                              &Global_Physical_Variables::C1,
                              &Global_Physical_Variables::E);
 // Define a constitutive law (based on strain energy function)
 Global_Physical_Variables::Constitutive_law_pt =
  new IsotropicStrainEnergyFunctionConstitutiveLaw(
   Global_Physical_Variables::Strain_energy_function_pt);
 //Set up the problem with pure displacement formulation
 ElasticFishProblem<RefineableQPVDElement<2,3> > problem;
 problem.run();


}
```

1.1 PDF file

A pdf version of this document is available.