

Chapter 1

Demo problem: Solution of a Poisson problem in an "elastic" domain

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

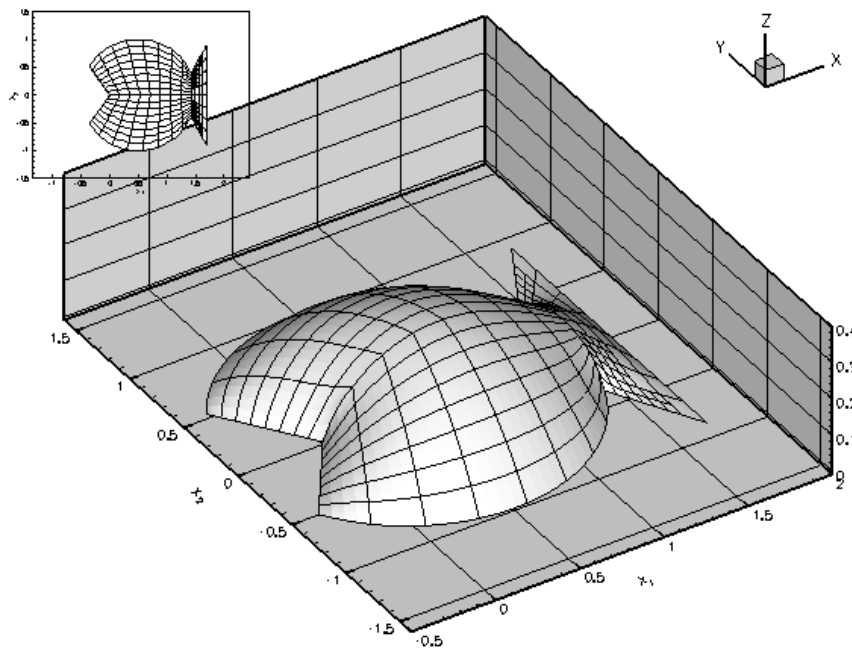


Figure 1.1 Adaptive solution of Poisson's equation in a fish-shaped domain for various 'widths' of the domain. The update of the nodal positions in response to changes in the boundary shape is done by pseudo-elasticity.

```
//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// Version 1.0; svn revision $LastChangedRevision$  
//LIC//  
//LIC// $LastChangedDate$  
//LIC//  
//LIC// Copyright (C) 2006-2016 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//=====
// Solve Poisson equation in deformable fish-shaped domain.
// Mesh deformation is driven by pseudo-elasticity approach.

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

// Poisson equations
#include "poisson.h"

// Solid mechanics
#include "solid.h"

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

// Circle as generalised element:
#include "circle_as_generalised_element.h"

using namespace std;
using namespace oomph;

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

//=====
// Namespace for const source term in Poisson equation
//=====
namespace ConstSourceForPoisson
{
    /// Strength of source function: default value 1.0
    double Strength=1.0;

    /// Const source function
    void get_source(const Vector<double>& x, double& source)
    {
        source = -Strength;
    }
}

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

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

    /// \short 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 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 curvilinear 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));
    }
}

/// 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 constitutive law
    ConstitutiveLaw* Constitutive_law_pt;

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

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

//=====
/// Solve Poisson equation on deforming fish-shaped domain.
/// Mesh update via pseudo-elasticity.
//=====
template<class ELEMENT>
class DeformableFishPoissonProblem : public Problem
{
public:

```

```

/// Constructor:
DeformableFishPoissonProblem();

/// Run simulation
void run();

/// Access function for the specific 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() {}

/// \short 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 solid 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:

/// Node at which the solution of the Poisson equation is documented
Node* Doc_node_pt;

/// Trace file
ofstream Trace_file;

// Geometric object/generalised element that represents the deformable
// fish back
ElasticallySupportedRingElement* Fish_back_pt;

};

//=====
/// Constructor:
//=====
template<class ELEMENT>
DeformableFishPoissonProblem<ELEMENT>::DeformableFishPoissonProblem
()
{
    // 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/generalised element that will become the
    // deformable fish back
    Fish_back_pt=new ElasticallySupportedRingElement(x_c,y_c,r_back);

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

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

    // Choose a node at which the solution is documented: Choose
    // the central node that is shared by all four elements in

```

```

// the base mesh because it exists at all refinement levels.

// How many nodes does element 0 have?
unsigned nnod=mesh_pt()->finite_element_pt(0)->nnode();

// The central node is the last node in element 0:
Doc_node_pt=mesh_pt()->finite_element_pt(0)->node_pt(nnod-1);

// Doc
cout << std::endl << "Control node is located at: "
      << Doc_node_pt->x(0) << " " << Doc_node_pt->x(1) << std::endl << std::endl;

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

// Change/doc targets for mesh adaptation
if (CommandLineArgs::Argc>1)
{
    mesh_pt()->max_permitted_error()=0.05;
    mesh_pt()->min_permitted_error()=0.005;
}
mesh_pt()->doc_adaptivity_targets(cout);

// Specify BC/source fct for Poisson problem:
//-----

// Set the Poisson boundary conditions for this problem: All nodes are
// free by default -- just pin the ones that have Dirichlet conditions
// here.
unsigned num_bound = mesh_pt()->nboundary();
for(unsigned ibound=0;ibound<num_bound;ibound++)
{
    unsigned num_nod=mesh_pt()->nboundary_node(ibound);
    for (unsigned inod=0;inod<num_nod;inod++)
    {
        mesh_pt()->boundary_node_pt(ibound,inod)->pin(0);
    }
}

// Set homogeneous boundary conditions for the Poisson equation
// on all boundaries
for(unsigned ibound=0;ibound<num_bound;ibound++)
{
    // Loop over the nodes on boundary
    unsigned num_nod=mesh_pt()->nboundary_node(ibound);
    for (unsigned inod=0;inod<num_nod;inod++)
    {
        mesh_pt()->boundary_node_pt(ibound,inod)->set_value(0,0.0);
    }
}

// Loop over elements and set pointers to source function
unsigned n_element = mesh_pt()->nelement();
for(unsigned i=0;i<n_element;i++)
{
    // Upcast from FiniteElement to the present element
    ELEMENT *el_pt = dynamic_cast<ELEMENT*>(mesh_pt()->element_pt(i));

    //Set the source function pointer
    el_pt->source_fct_pt() = &ConstSourceForPoisson::get_source;
}

// Specify BC/source fct etc for (pseudo-)Solid problem
//-----

// Pin all nodal positions
for(unsigned ibound=0;ibound<num_bound;ibound++)
{
    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 Solid parameters/function pointers
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;
}

// 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 DeformableFishPoissonProblem<ELEMENT>::doc_solution
    (DocInfo& doc_info)
{
    ofstream some_file;
    char filename[100];

    // Number of plot points
    unsigned npts = 5;

    // Call output function for all elements
    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();

    // Write vertical position of the fish back, and solution at
    // control node to trace file
    Trace_file
        << static_cast<Circle*>(mesh_pt()->fish_back_pt())->y_c()
        << " " << Doc_node_pt->value(0) << std::endl;
}

//=====
/// Run the problem
//=====
template<class ELEMENT>
void DeformableFishPoissonProblem<ELEMENT>::run()
{
    // Output
    DocInfo doc_info;

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

    // Step number
    doc_info.number()=0;

    // Open trace file
    char filename[100];
    sprintf(filename,"%s/trace.dat",doc_info.directory().c_str());
    Trace_file.open(filename);

    Trace_file << "VARIABLES=\<sub>circle</sub>\", \"\<sub>control</sub>\" \"
        << std::endl;

    //Parameter incrementation
    unsigned nstep=5;
    for(unsigned i=0;i<nstep;i++)

```

```

{
    //Solve the problem with Newton's method, allowing for up to 2
    //rounds of adaptation
    newton_solve(2);

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

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

}

//=====
/// Driver for simple elastic problem.
/// If there are any command line arguments, we regard this as a
/// validation run and perform only a single step.
//=====
int main(int argc, char* argv[])
{
    // Store command line arguments
    CommandLineArgs::setup(argc,argv);

    //Set physical parameters
    Global_Physical_Variables::Nu = 0.4;

    // Define a constitutive law (based on strain energy function)
    Global_Physical_Variables::Constitutive_law_pt =
        new GeneralisedHookean(&Global_Physical_Variables::Nu);

    // Set up the problem: Choose a hybrid element that combines the
    // 3x3 node refineable quad Poisson element with a displacement-based
    // solid-mechanics element (for the pseudo-elastic mesh update in response
    // to changes in the boundary shape)
    DeformableFishPoissonProblem<
        RefineablePseudoSolidNodeUpdateElement<RefineableQPoissonElement<2,3>,
        RefineableQPVDElement<2,3> >
        > problem;

    problem.run();
}

```

1.1 PDF file

A [pdf version](#) of this document is available.