

Chapter 1

A real fluid-structure interaction problem: Finite Reynolds number flow in an oscillating elastic ring.

Our first "real" fluid-structure interaction problem: We study the finite-Reynolds number internal flow generated by the motion of an oscillating elastic ring and compare the results against asymptotic predictions.

```
//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 2D Navier Stokes flow interacting with an elastic ring
// Oomph-lib include files
#include "generic.h"
#include "navier_stokes.h"
#include "beam.h"
//Need to include templated meshes, so that all functions
//are instantiated for our particular element types.
#include "meshes/quarter_circle_sector_mesh.h"
#include "meshes/one_d_lagrangian_mesh.h"
using namespace std;
using namespace oomph;
//=====
/// Namespace for physical parameters
//=====
namespace Global_Physical_Variables
{
  // Independent parameters:
  //-----

  /// Square of Womersly number (a frequency parameter)
  double Alpha_sq=50.0;

  /// Density ratio of the solid and the fluid
  double Density_ratio=1.0;

  /// External Pressure
  double Pext=0.0;

  /// Poisson ratio
```

```

double Nu=0.49;

/// Nondimensional thickness of the beam
double H=0.05;

/// Perturbation pressure
double Pcos=0.0;
/// Dependent parameters:
//-----

/// Reynolds number
double Re;

/// Reynolds x Strouhal number
double ReSt;

/// Timescale ratio (non-dimensation density)
double Lambda_sq;

/// Stress ratio
double Q;

/// Set the parameters that are used in the code as a function
/// of the Womersley number, the density ratio and H
void set_params()
{
    cout << "\n\n===== " <<std::endl;
    cout << "\nSetting parameters. \n\n";
    cout << "Prescribed: Square of Womersley number: Alpha_sq = "
    << Alpha_sq << std::endl;
    cout << "          Density ratio:          Density_ratio = "
    << Density_ratio << std::endl;
    cout << "          Wall thickness:              H = "
    << H << std::endl;
    cout << "          Poisson ratio:              Nu = "
    << Nu << std::endl;
    cout << "          Pressure perturbation:      Pcos = "
    << Pcos << std::endl;
    Q=1.0/12.0*pow(H,3)/Alpha_sq;
    cout << "\nDependent: Stress ratio:          Q = "
    << Q << std::endl;
    Lambda_sq=1.0/12.0*pow(H,3)*Density_ratio;
    cout << "          Timescale ratio:          Lambda_sq = "
    << Lambda_sq << std::endl;
    Re=Alpha_sq;
    cout << "          Reynolds number:          Re = "
    << Re << std::endl;
    ReSt=Re;
    cout << "          Womersley number:          ReSt = "
    << ReSt << std::endl;
    cout << "\n\n===== \n\n"
    <<std::endl;
}

/// Non-FSI load function, a constant external pressure plus
/// a (small) sinusoidal perturbation of wavenumber two.
void pcos_load(const Vector<double>& xi, const Vector<double> &x,
               const Vector<double>& N, Vector<double>& load)
{
    for(unsigned i=0;i<2;i++)
    {load[i] = (Pext - Pcos*cos(2.0*xi[0]))*N[i];}
}
}
//=====
/// FSI Ring problem: a fluid-structure interaction problem in which
/// a viscous fluid bounded by an initially circular beam is set into motion
/// by a small sinusoidal perturbation of the beam (the domain boundary).
//=====
class FSIRingProblem : public Problem
{
    /// There are very few element types that will work for this problem.
    /// Rather than passing the element type as a template parameter to the
    /// problem, we choose instead to use a typedef to specify the
    /// particular element fluid used.
    typedef AlgebraicElement<RefineableQCrouzeixRaviartElement<2> > FLUID_ELEMENT;

    /// Typedef to specify the solid element used
    typedef FSIHermiteBeamElement SOLID_ELEMENT;
public:

    /// Constructor: Number of elements in wall mesh, amplitude of the
    /// initial wall deformation, amplitude of pcos perturbation and its duration.
    FSIRingProblem(const unsigned &nelement_wall,
                   const double& eps_ampl, const double& pcos_initial,
                   const double& pcos_duration);

    /// Update after solve (empty)

```

```

void actions_after_newton_solve() {}

/// Update before solve (empty)
void actions_before_newton_solve() {}

/// Update the problem specs before checking Newton
/// convergence
void actions_before_newton_convergence_check()
{
    // Update the fluid mesh -- auxiliary update function for algebraic
    // nodes automatically updates no slip condition.
    Fluid_mesh_pt->node_update();
}

/// Update the problem specs after adaptation:
void actions_after_adapt()
{
    // The functions used to update the no slip boundary conditions
    // must be set on any new nodes that have been created during the
    // mesh adaptation process.
    // There is no mechanism by which auxiliary update functions
    // are copied to newly created nodes.
    // (because, unlike boundary conditions, they don't occur exclusively
    // at boundaries)
    // The no-slip boundary is boundary 1 of the mesh
    // Loop over the nodes on this boundary and reset the auxilliary
    // node update function
    unsigned n_node = Fluid_mesh_pt->nboundary_node(1);
    for (unsigned n=0; n<n_node; n++)
    {
        Fluid_mesh_pt->boundary_node_pt(1,n)->set_auxiliary_node_update_fct_pt(
            FSI_functions::apply_no_slip_on_moving_wall);
    }
    // (Re-)setup fsi: Work out which fluid dofs affect wall elements
    // the correspondance between wall dofs and fluid elements is handled
    // during the remeshing, but the "reverse" association must be done
    // separately.
    // We need to set up the interaction every time because the fluid element
    // adjacent to a given solid element's integration point may have changed
    // We pass the boundary between the fluid and solid meshes and pointers
    // to the meshes. The interaction boundary is boundary 1 of the 2D
    // fluid mesh.
    FSI_functions::setup_fluid_load_info_for_solid_elements<FLUID_ELEMENT,2>
        (this,1,Fluid_mesh_pt,Wall_mesh_pt);
}

/// Doc solution: Pass number of timestep, i (we append to tracefile
/// after every timestep but do a full doc only at certain intervals),
/// DocInfo object and tracefile
void doc_solution(const unsigned& i, DocInfo& doc_info, ofstream& trace_file);

/// Do dynamic run
void dynamic_run();
private:

/// Setup initial condition for both domains
void set_initial_condition();

/// Setup initial condition for wall
void set_wall_initial_condition();

/// Setup initial condition for fluid
void set_fluid_initial_condition();

/// Element used for documenting displacement
SOLID_ELEMENT* Doc_displacement_elem_pt;

/// Pointer to wall mesh
OneDLagrangianMesh<SOLID_ELEMENT> *Wall_mesh_pt;

/// Pointer to fluid mesh
AlgebraicRefineableQuarterCircleSectorMesh<FLUID_ELEMENT> *Fluid_mesh_pt;

/// Pointer to geometric object that represents the undeformed wall shape
GeomObject* Undef_geom_pt;

/// Pointer to wall timestepper
Newmark<2>* Wall_time_stepper_pt;

/// Pointer to fluid timestepper
BDF<2>* Fluid_time_stepper_pt;

/// Pointer to node on coarsest mesh on which velocity is traced
Node* Veloc_trace_node_pt;

/// Amplitude of initial deformation
double Eps_ampl;

```

```

/// Initial pcos
double Pcos_initial;

/// Duration of initial pcos
double Pcos_duration;
};
//=====
/// Setup initial condition: When we're done here, all variables
/// represent the state at the initial time.
//=====
void FSIRingProblem::set_initial_condition()
{
    cout << "Setting wall ic" << std::endl;
    set_wall_initial_condition();
    cout << "Setting fluid ic" << std::endl;
    set_fluid_initial_condition();
}
//=====
/// Setup initial condition for fluid: Impulsive start
//=====
void FSIRingProblem::set_fluid_initial_condition()
{
    // Update fluid domain: Careful!!! This also applies the no slip conditions
    // on all nodes on the wall! Since the wall might have moved since
    // we created the mesh; we're therefore imposing a nonzero
    // velocity on these nodes. Must wipe this afterwards (done
    // by setting *all* velocities to zero) otherwise we get
    // an impulsive start from a very bizarre initial velocity
    // field! [Yes, it took me a while to figure this out...]
    Fluid_mesh_pt->node_update();
    // Assign initial values for the velocities;
    // pressures don't carry a time history and can be left alone.
    // Find number of nodes in fluid mesh
    unsigned n_node = Fluid_mesh_pt->nnode();
    // Loop over the nodes to set initial guess everywhere
    for(unsigned n=0; n<n_node; n++)
    {
        // Loop over velocity directions: Impulsive initial start from
        // zero velocity!
        for(unsigned i=0; i<2; i++)
        {
            Fluid_mesh_pt->node_pt(n)->set_value(i, 0.0);
        }
    }
    // Do an impulsive start with the assigned velocity field
    Fluid_mesh_pt->assign_initial_values_impulsive();
}
//=====
/// Setup initial condition: Impulsive start either from
/// deformed or undeformed wall shape.
//=====
void FSIRingProblem::set_wall_initial_condition()
{
    // Geometric object that specifies the initial conditions:
    // A ring that is buckled in a 2-lobed mode
    GeomObject* ic_geom_object_pt =
        new PseudoBucklingRing(Eps_ampl, Global_Physical_Variables::H, 2, 2,
                               Wall_time_stepper_pt);

    // Assign period of oscillation of the geometric object
    static_cast<PseudoBucklingRing*>(ic_geom_object_pt)->set_T(1.0);

    // Set initial time (to deform wall into max. amplitude)
    double time=0.25;

    // Assign initial radius of the object
    static_cast<PseudoBucklingRing*>(ic_geom_object_pt)->set_R_0(1.00);

    // Setup object that specifies the initial conditions:
    SolidInitialCondition* IC_pt = new SolidInitialCondition(ic_geom_object_pt);

    // Assign values of positional data of all elements on wall mesh
    // so that the wall deforms into the shape specified by IC object.
    SolidMesh::Solid_IC_problem.set_static_initial_condition(
        this, Wall_mesh_pt, IC_pt, time);
}
//=====
/// Document solution: Pass number of timestep, i; we append to trace file
/// at every timestep and do a full doc only after a certain number
/// of steps.
//=====
void FSIRingProblem::doc_solution(const unsigned& i,
    DocInfo& doc_info, ofstream& trace_file)
{
    // Full doc every nskip steps

```

```

unsigned nskip=1; // ADJUST

// If we at an integer multiple of nskip, full documentation.
if (i%nskip==0)
{
    doc_info.enable_doc();
    cout << "Full doc step " << doc_info.number()
         << " for time " << time_stepper_pt()->time_pt()->time() << std::endl;
}
//Otherwise, just output the trace file
else
{
    doc_info.disable_doc();
    cout << "Only trace for time "
         << time_stepper_pt()->time_pt()->time() << std::endl;
}

// If we are at a full documentation step, output the fluid solution
if (doc_info.is_doc_enabled())
{
    //Variables used in the output file.
    ofstream some_file; char filename[100];
    //Construct the output filename from the doc_info number and the
    //output directory
    sprintf(filename,"%s/soln%i.dat",doc_info.directory().c_str(),
            doc_info.number());
    //Open the output file
    some_file.open(filename);
    /// Output the solution using 5x5 plot points
    Fluid_mesh_pt->output(some_file,5);
    //Close the output file
    some_file.close();
}

//Temporary vector to give the local coordinate at which to document
//the wall displacement
Vector<double> s(1,1.0);
// Write to the trace file:
trace_file << time_pt()->time()
//Document the displacement at the end of the the chosen element
<< " " << Doc_displacement_elem_pt->interpolated_x(s,1)
<< " " << Veloc_trace_node_pt->x(0)
<< " " << Veloc_trace_node_pt->x(1)
<< " " << Veloc_trace_node_pt->value(0)
<< " " << Veloc_trace_node_pt->value(1)
<< " " << Fluid_mesh_pt->nelement()
<< " " << ndof()
<< " " << Fluid_mesh_pt->nrefinement_overruled()
<< " " << Fluid_mesh_pt->max_error()
<< " " << Fluid_mesh_pt->min_error()
<< " " << Fluid_mesh_pt->max_permitted_error()
<< " " << Fluid_mesh_pt->min_permitted_error()
<< " " << Fluid_mesh_pt->max_keep_unrefined();
// Output the number of the corresponding full documentation
// file number (or -1 if no full doc was made)
if (doc_info.is_doc_enabled())
{trace_file << " " << doc_info.number() << " " ;}
else {trace_file << " " << -1 << " " ;}

//End the trace file
trace_file << std::endl;

// Increment counter for full doc
if (doc_info.is_doc_enabled()) {doc_info.number()++;}
}
//=====
/// Constructor for FSI ring problem. Pass number of wall elements
/// and length of wall (in Lagrangian coordinates) amplitude of
/// initial deformation, pcos perturbation and duration.
//=====
FSIRingProblem::FSIRingProblem(const unsigned& N,
                                const double& eps_ampl, const double& pcos_initial,
                                const double& pcos_duration) :
Eps_ampl(eps_ampl), Pcos_initial(pcos_initial),
Pcos_duration(pcos_duration)
{
    //-----
    // Create timesteppers
    //-----

    // Allocate the wall timestepper and add it to the problem's vector
    // of timesteppers
    Wall_time_stepper_pt = new Newmark<2>;
    add_time_stepper_pt(Wall_time_stepper_pt);
    // Allocate the fluid timestepper and add it to the problem's Vector
    // of timesteppers

```

```

Fluid_time_stepper_pt = new BDF<2>;
add_time_stepper_pt(Fluid_time_stepper_pt);
//-----
// Create the wall mesh
//-----
// Undeformed wall is an elliptical ring
Undef_geom_pt = new Ellipse(1.0,1.0);
//Length of wall in Lagrangian coordinates
double L = 2.0*atan(1.0);
//Now create the (Lagrangian!) mesh
Wall_mesh_pt = new
OneDLagrangianMesh<SOLID_ELEMENT>(N,L,Undef_geom_pt,Wall_time_stepper_pt);
//-----
// Set the boundary conditions for wall mesh (problem)
//-----

// Bottom boundary: (Boundary 0)
// No vertical displacement
Wall_mesh_pt->boundary_node_pt(0,0)->pin_position(1);
// Zero slope: Pin type 1 dof for displacement direction 0
Wall_mesh_pt->boundary_node_pt(0,0)->pin_position(1,0);

// Top boundary: (Boundary 1)
// No horizontal displacement
Wall_mesh_pt->boundary_node_pt(1,0)->pin_position(0);
// Zero slope: Pin type 1 dof for displacement direction 1
Wall_mesh_pt->boundary_node_pt(1,0)->pin_position(1,1);
//-----
// Create the fluid mesh:
//-----
// Fluid mesh is suspended from wall between the following Lagrangian
// coordinates:
double xi_lo=0.0;
double xi_hi=L;
// Fractional position of dividing line for two outer blocks in mesh
double fract_mid=0.5;

//Create a geometric object that represents the wall geometry from the
//wall mesh (one Lagrangian, two Eulerian coordinates).
MeshAsGeomObject *wall_mesh_as_geometric_object_pt
= new MeshAsGeomObject(Wall_mesh_pt);
// Build fluid mesh using the wall mesh as a geometric object
Fluid_mesh_pt = new AlgebraicRefineableQuarterCircleSectorMesh<FLUID_ELEMENT >
(wall_mesh_as_geometric_object_pt,
xi_lo,fract_mid,xi_hi,Fluid_time_stepper_pt);
// Set the error estimator
Z2ErrorEstimator* error_estimator_pt=new Z2ErrorEstimator;
Fluid_mesh_pt->spatial_error_estimator_pt()=error_estimator_pt;

// Extract pointer to node at center of mesh
unsigned nnode=Fluid_mesh_pt->finite_element_pt(0)->nnode();
Veloc_trace_node_pt=Fluid_mesh_pt->finite_element_pt(0)->node_pt(nnode-1);

//-----
// Set the fluid boundary conditions
//-----
// Bottom boundary (boundary 0):
{
    unsigned n_node = Fluid_mesh_pt->nboundary_node(0);
    for (unsigned n=0;n<n_node;n++)
    {
        // Pin vertical velocity
        Fluid_mesh_pt->boundary_node_pt(0,n)->pin(1);
    }
}

// Ring boundary (boundary 1):
// No slip; this also implies that the velocity needs
// to be updated in response to wall motion
{
    unsigned n_node = Fluid_mesh_pt->nboundary_node(1);
    for (unsigned n=0;n<n_node;n++)
    {
        // Which node are we dealing with?
        Node* node_pt=Fluid_mesh_pt->boundary_node_pt(1,n);
        // Set auxiliary update function pointer
        node_pt->set_auxiliary_node_update_fct_pt(
            FSI_functions::apply_no_slip_on_moving_wall);

        // Pin both velocities
        for(unsigned i=0;i<2;i++) {node_pt->pin(i);}
    }
}
// Left boundary (boundary 2):
{
    unsigned n_node = Fluid_mesh_pt->nboundary_node(2);
    for (unsigned n=0;n<n_node;n++)

```

```

    {
        // Pin horizontal velocity
        Fluid_mesh_pt->boundary_node_pt(2,n)->pin(0);
    }
}
//-----
// Add the submeshes and build global mesh
// -----
// Wall mesh
add_sub_mesh(Wall_mesh_pt);
//Fluid mesh
add_sub_mesh(Fluid_mesh_pt);
// Combine all submeshes into a single Mesh
build_global_mesh();

//-----
// Finish problem setup
// -----
//Find number of elements in fluid mesh
unsigned n_element = Fluid_mesh_pt->nelement();
// Loop over the fluid elements to set up element-specific
// things that cannot be handled by constructor
for(unsigned e=0;e<n_element;e++)
{
    // Upcast from FiniteElement to the present element
    FLUID_ELEMENT *el_pt
    = dynamic_cast<FLUID_ELEMENT*>(Fluid_mesh_pt->element_pt(e));
    //Set the Reynolds number, etc
    el_pt->re_pt() = &Global_Physical_Variables::Re;
    el_pt->re_st_pt() = &Global_Physical_Variables::ReSt;
    el_pt->evaluate_shape_derivs_by_direct_fd();
    // el_pt->evaluate_shape_derivs_by_chain_rule();
    // el_pt->enable_always_evaluate_dresidual_dnodal_coordinates_by_fd();
    // if (e==0)
    // {
    //     el_pt->disable_always_evaluate_dresidual_dnodal_coordinates_by_fd();
    // }
    // else
    // {
    //     el_pt->enable_always_evaluate_dresidual_dnodal_coordinates_by_fd();
    // }
    //el_pt->evaluate_shape_derivs_by_direct_fd();
}

//Loop over the solid elements to set physical parameters etc.
unsigned n_wall_element = Wall_mesh_pt->nelement();
for(unsigned e=0;e<n_wall_element;e++)
{
    //Cast to proper element type
    SOLID_ELEMENT *el_pt = dynamic_cast<SOLID_ELEMENT*>(
        Wall_mesh_pt->element_pt(e));

    //Set physical parameters for each element:
    el_pt->h_pt() = &Global_Physical_Variables::H;
    el_pt->lambda_sq_pt() = &Global_Physical_Variables::Lambda_sq;

    //Function that specifies the external load Vector
    el_pt->load_vector_fct_pt() = &Global_Physical_Variables::pcos_load;
    // Function that specifies the load ratios
    el_pt->q_pt() = &Global_Physical_Variables::Q;
    //Assign the undeformed beam shape
    el_pt->undeformed_beam_pt() = Undef_geom_pt;
}

// Establish control displacement: (even if no displacement control is applied
// we still want to doc the displacement at the same point)
// Choose element: (This is the last one)
Doc_displacement_elem_pt = dynamic_cast<SOLID_ELEMENT*>(
    Wall_mesh_pt->element_pt(n_wall_element-1));

// Setup fsi: Work out which fluid dofs affect the wall elements
// the correspondance between wall dofs and fluid elements is handled
// during the remeshing, but the "reverse" association must be done
// separately.
// We pass the boundary between the fluid and solid meshes and pointers
// to the meshes. The interaction boundary is boundary 1 of the
// 2D fluid mesh.
FSI_functions::setup_fluid_load_info_for_solid_elements<FLUID_ELEMENT,2>
    (this,1,Fluid_mesh_pt,Wall_mesh_pt);
// Do equation numbering
cout << " # of dofs " << assign_eqn_numbers() << std::endl;
}
//=====
/// Solver loop to perform unsteady run
//=====

```

```

void FSIRingProblem::dynamic_run()
{
    // Setup documentation
    //-----

    /// Label for output
    DocInfo doc_info;
    // Output directory
    doc_info.set_directory("RESLT");
    // Step number
    doc_info.number()=0;
    //Open a trace file
    ofstream trace_file("RESLT/trace_ring.dat");

    // Write header for trace file
    trace_file << "VARIABLES=\"time\", \"V_c_t_r_l\"";
    trace_file << ", \"x<sub>1</sub><sup>(track)</sup>\"";
    trace_file << ", \"x<sub>2</sub><sup>(track)</sup>\"";
    trace_file << ", \"u<sub>1</sub><sup>(track)</sup>\"";
    trace_file << ", \"u<sub>2</sub><sup>(track)</sup>\"";
    trace_file << ", \"N<sub>element</sub>\"";
    trace_file << ", \"N<sub>dof</sub>\"";
    trace_file << ", \"# of under-refined elements\"";
    trace_file << ", \"max. error\"";
    trace_file << ", \"min. error\"";
    trace_file << ", \"max. permitted error\"";
    trace_file << ", \"min. permitted error\"";
    trace_file << ", \"max. permitted # of unrefined elements\"";
    trace_file << ", \"doc number\"";
    trace_file << std::endl;

    // Initialise timestepping
    // -----
    // Number of steps
    unsigned nstep=300;
    // Nontrivial command line input: validation: only do three steps
    if (CommandLineArgs::Argc>1)
    {
        nstep=1;
        cout << "Only doing nstep steps for validation: " << nstep << std::endl;
    }
    // Set initial timestep
    double dt=0.004;
    // Set initial value for dt -- also assigns weights to the timesteppers
    initialise_dt(dt);
    // Set physical parameters
    //-----
    using namespace Global_Physical_Variables;
    // Set Womersley number
    Alpha_sq=100.0; // 50.0; // ADJUST
    // Set density ratio
    Density_ratio=10.0; // 0.0; ADJUST
    // Wall thickness
    H=1.0/20.0;
    // Set external pressure
    Pext=0.0;

    /// Perturbation pressure
    Pcos=Pcos_initial;
    // Assign/doc corresponding computational parameters
    set_params();
    // Refine uniformly and assign initial conditions
    //-----
    // Refine the problem uniformly
    refine_uniformly();
    refine_uniformly();
    // This sets up the solution at the initial time
    set_initial_condition();
    // Set targets for spatial adptivity
    //-----
    // Max. and min. error for adaptive refinement/unrefinement
    Fluid_mesh_pt->max_permitted_error()=1.0e-2;
    Fluid_mesh_pt->min_permitted_error()=1.0e-3;
    // Don't allow refinement to drop under given level
    Fluid_mesh_pt->min_refinement_level()=2;
    // Don't allow refinement beyond given level
    Fluid_mesh_pt->max_refinement_level()=6;
    // Don't bother adapting the mesh if no refinement is required
    // and if less than ... elements are to be merged.
    Fluid_mesh_pt->max_keep_unrefined()=20;
    // Doc refinement targets
    Fluid_mesh_pt->doc_adaptivity_targets(cout);
    // Do the timestepping
    //-----
    // Reset initial conditions after refinement for first step only
    bool first=true;
    //Output initial data

```



```

doc_solution(0,doc_info,trace_file);
// {
//   unsigned nel=Fluid_mesh_pt->nelement();
//   for (unsigned e=0;e<nel;e++)
//   {
//     std::cout << "\n\nEl: " << e << std::endl << std::endl;
//     FiniteElement* el_pt=Fluid_mesh_pt->finite_element_pt(e);
//     unsigned n_dof=el_pt->ndof();
//     Vector<double> residuals(n_dof);
//     DenseDoubleMatrix jacobian(n_dof,n_dof);
//     el_pt->get_jacobian(residuals,jacobian);
//   }
//   exit(0);
// }
// Time integration loop
for(unsigned i=1;i<=nstep;i++)
{
  // Switch doc off during solve
  doc_info.disable_doc();
  // Solve
  unsigned max_adapt=1;
  unsteady_newton_solve(dt,max_adapt,first);
  // Now we've done the first step
  first=false;

  // Doc solution
  doc_solution(i,doc_info,trace_file);

  /// Switch off perturbation pressure
  if (time_pt()->time()>Pcos_duration) {Pcos=0.0;}
}
}
//=====
/// Driver for fsi ring test problem
//=====
int main(int argc, char* argv[])
{
  // Store command line arguments
  CommandLineArgs::setup(argc,argv);

  // Number of elements
  unsigned nelelem = 13;

  /// Perturbation pressure
  double pcos_initial=1.0e-6; // ADJUST

  /// Duration of initial pcos perturbation
  double pcos_duration=0.3; // ADJUST

  /// Amplitude of initial deformation
  double eps_ampl=0.0; // ADJUST
  //Set up the problem
  FSIRingProblem problem(nelelem,eps_ampl,pcos_initial,pcos_duration);
  // Do parameter study
  problem.dynamic_run();
}

```

1.1 PDF file

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