

## Chapter 1

# The spatially-adaptive solution of the azimuthally Fourier-decomposed equations of 3D time-harmonic linear elasticity on unstructured meshes

In this tutorial we re-visit the solution of the time-harmonic equations of 3D linear elasticity in cylindrical polar coordinates, using a Fourier decomposition of the solution in the azimuthal direction. The driver code is very similar to the one discussed in [another tutorial](#) – the main purpose of the current tutorial is to demonstrate the use of spatial adaptivity on unstructured meshes. Compared to the test case considered in the [other tutorial](#) we study a slightly less contrived test problem: the forced time-harmonic oscillations of a finite-length, hollow cylinder, loaded by a time-periodic pressure load on its inner surface.

**Acknowledgement:** This implementation of the equations and the documentation were developed jointly with Robert Harter (Thales Underwater Systems Ltd) with financial support from a KTA Secondment grant from University of Manchester's EPSRC-funded Knowledge Transfer Account.

---

### 1.1 The test problem

The figure below shows the problem considered in this tutorial: an annular elastic body that occupies the region  $r_{\min} \leq r \leq r_{\max}, z_{\min} \leq z \leq z_{\max}$  is loaded by a time-harmonic pressure load acting on its inner surface (at  $r = r_{\min}$ ). The upper and lower ends of the hollow cylinder (at  $z = z_{\min}$  and  $z = z_{\max}$ ) are held at a fixed position. Here is an animation of the resulting displacement field for  $r_{\min} = 0.1$  and  $r_{\max} = 1.1$ .



Figure 1.1 Forced oscillations of a thick-walled, hollow cylinder, subject to a pressure load on its inner surface. The pink shape in the background shows the cylinder's undeformed shape (in a radial plane); the mesh plotted in the region  $r < 0$  illustrates how spatial adaptivity refines the mesh in regions of sharp displacement gradients (near the loaded surface and the supports).

## 1.2 The numerical solution

The driver code for this problem is very similar to the one discussed in [another tutorial](#). Running `sdiff` on the two driver codes

```
demo_drivers/time_harmonic_fourier_decomposed_linear_↵
elasticity/cylinder/cylinder.cc
```

and

```
demo_drivers/time_harmonic_fourier_decomposed_linear_↵
elasticity/cylinder/pressure_loaded_cylinder.cc
```

shows you the differences, the most important of which are:

- The change of the forcing to a spatially constant pressure load on the inside boundary.
- The provision of the `actions_before/after_adapt()` functions and a helper function `complete_↵ problem_setup()` which rebuilds the elements (by passing the problem parameters to the elements) following the unstructured mesh adaptation. (The need/rationale for such a function is discussed in [another tutorial](#).)
- The mesh generation and the application of boundary conditions at the upper and lower boundaries of the hollow cylinder.

All of this is reasonably straightforward and provides a powerful code that automatically adapts the mesh in regions of large displacement gradients. Have a look through the driver code and play with it.

## 1.3 Code listing

Here's a listing of the complete driver code:

```

//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) 2006-2021 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
// The oomphlib headers
#include "generic.h"
#include "time_harmonic_fourier_decomposed_linear_elasticity.h"
// The mesh
#include "meshes/rectangular_quadmesh.h"
#include "meshes/triangle_mesh.h"
using namespace std;
using namespace oomph;
//===start_of_namespace=====
// Namespace for global parameters
//=====
namespace Global_Parameters
{
  // Define Poisson's ratio Nu
  std::complex<double> Nu(0.3,0.0);

  // Define the non-dimensional Young's modulus
  std::complex<double> E(1.0,0.0);

  // Define Fourier wavenumber
  int Fourier_wavenumber = 0;

  // \short Define the non-dimensional square angular frequency of
  // time-harmonic motion
  std::complex<double> Omega_sq (10.0,0.0);

  // Length of domain in r direction
  double Lr = 1.0;

  // Length of domain in z-direction
  double Lz = 2.0;
  // Set up min & max (r,z) coordinates
  double rmin = 0.1;
  double zmin = 0.3;
  double rmax = rmin+Lr;
  double zmax = zmin+Lz;

  // Define the imaginary unit
  const std::complex<double> I(0.0,1.0);
  // Pressure load
  double P=1.0;

  // The traction function at r=rmin: (t_r, t_z, t_theta)
  void boundary_traction(const Vector<double> &x,
                        const Vector<double> &n,
                        Vector<std::complex<double> > &result)
  {
    // Radial traction
    result[0] = P;
    // Axial traction
    result[1] = 0.0;
    // Azimuthal traction
    result[2] = 0.0;
  }

} // end_of_namespace
//===start_of_problem_class=====
// Class to validate time harmonic linear elasticity (Fourier
// decomposed)
//=====

```

```

template<class ELEMENT>
class FourierDecomposedTimeHarmonicLinearElasticityProblem : public Problem
{
public:

    /// \short Constructor: Pass number of elements in r and z directions
    /// and boundary locations
    FourierDecomposedTimeHarmonicLinearElasticityProblem(
        const unsigned &nr, const unsigned &nz,
        const double &rmin, const double& rmax,
        const double &zmin, const double& zmax);

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

    /// Update after solve is empty
    void actions_after_newton_solve() {}

    /// Delete traction elements
    void delete_traction_elements();

    /// Helper function to complete problem setup
    void complete_problem_setup();

    /// Actions before adapt: Wipe the mesh of traction elements
    void actions_before_adapt()
    {
        // Kill the traction elements and wipe surface mesh
        delete_traction_elements();

        // Rebuild the Problem's global mesh from its various sub-meshes
        rebuild_global_mesh();
    }

    /// Actions after adapt: Rebuild the mesh of traction elements
    void actions_after_adapt()
    {
        // Create traction elements from all elements that are
        // adjacent to FSI boundaries and add them to surface meshes
        assign_traction_elements();

        // Rebuild the Problem's global mesh from its various sub-meshes
        rebuild_global_mesh();

        // Complete problem setup
        complete_problem_setup();
    }

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

private:

    /// Allocate traction elements on the bottom surface
    void assign_traction_elements();

#ifdef ADAPTIVE

    /// Pointer to the bulk mesh
    RefineableTriangleMesh<ELEMENT>* Bulk_mesh_pt;
#else

    /// Pointer to the bulk mesh
    Mesh* Bulk_mesh_pt;
#endif

    /// Pointer to the mesh of traction elements
    Mesh* Surface_mesh_pt;
}; // end_of_problem_class
//===start_of_constructor=====
/// Problem constructor: Pass number of elements in coordinate
/// directions and size of domain.
//========
template<class ELEMENT>
FourierDecomposedTimeHarmonicLinearElasticityProblem<ELEMENT>::
FourierDecomposedTimeHarmonicLinearElasticityProblem
(const unsigned &nr, const unsigned &nz,
 const double &rmin, const double& rmax,
 const double &zmin, const double& zmax)
{
#ifdef ADAPTIVE
    // The boundary is bounded by four distinct boundaries, each
    // represented by its own polyline
    Vector<TriangleMeshCurveSection*> boundary_polyline_pt(4);

    // Vertex coordinates on boundary
    Vector<Vector<double>> > bound_coords(2);

```

```

bound_coords[0].resize(2);
bound_coords[1].resize(2);

// Horizontal bottom boundary
bound_coords[0][0]=rmin;
bound_coords[0][1]=zmin;
bound_coords[1][0]=rmax;
bound_coords[1][1]=zmin;

// Build the boundary polyline
unsigned boundary_id=0;
boundary_polyline_pt[0]=new TriangleMeshPolyLine(bound_coords,boundary_id);

// Vertical outer boundary
bound_coords[0][0]=rmax;
bound_coords[0][1]=zmin;
bound_coords[1][0]=rmax;
bound_coords[1][1]=zmax;

// Build the boundary polyline
boundary_id=1;
boundary_polyline_pt[1]=new TriangleMeshPolyLine(bound_coords,boundary_id);

// Horizontal top boundary
bound_coords[0][0]=rmax;
bound_coords[0][1]=zmax;
bound_coords[1][0]=rmin;
bound_coords[1][1]=zmax;

// Build the boundary polyline
boundary_id=2;
boundary_polyline_pt[2]=new TriangleMeshPolyLine(bound_coords,boundary_id);

// Vertical inner boundary
bound_coords[0][0]=rmin;
bound_coords[0][1]=zmax;
bound_coords[1][0]=rmin;
bound_coords[1][1]=zmin;

// Build the boundary polyline
boundary_id=3;
boundary_polyline_pt[3]=new TriangleMeshPolyLine(bound_coords,boundary_id);

// Pointer to the closed curve that defines the outer boundary
TriangleMeshClosedCurve* closed_curve_pt=
    new TriangleMeshPolygon(boundary_polyline_pt);

// Use the TriangleMeshParameters object for helping on the manage of the
// TriangleMesh parameters
TriangleMeshParameters triangle_mesh_parameters(closed_curve_pt);
// Specify the maximum area element
double uniform_element_area=0.2;
triangle_mesh_parameters.element_area() = uniform_element_area;
// Create the mesh
Bulk_mesh_pt=new RefineableTriangleMesh<ELEMENT>(triangle_mesh_parameters);
// Set error estimator
Bulk_mesh_pt->spatial_error_estimator_pt()=new Z2ErrorEstimator;
#else
//Now create the mesh
Bulk_mesh_pt = new RectangularQuadMesh<ELEMENT>(nr,nz,rmin,rmax,zmin,zmax);
#endif
//Create the surface mesh of traction elements
Surface_mesh_pt=new Mesh;
assign_traction_elements();

// Complete problem setup
complete_problem_setup();
// Add the submeshes to the problem
add_sub_mesh(Bulk_mesh_pt);
add_sub_mesh(Surface_mesh_pt);
// Now build the global mesh
build_global_mesh();
// Assign equation numbers
cout << assign_eqn_numbers() << " equations assigned" << std::endl;
} // end of constructor
//===start_of_complete_problem_setup=====
// Complete problem setup
//=====
template<class ELEMENT>
void FourierDecomposedTimeHarmonicLinearElasticityProblem<ELEMENT>::
complete_problem_setup()
{
    // Set the boundary conditions for this problem: All nodes are
    // free by default -- just pin & set the ones that have Dirichlet
    // conditions here
    // Pin displacements everywhere apart from boundaries 1 and 3
    //-----

```

```

for (unsigned ibound=0;ibound<3;ibound=ibound+2)
{
    unsigned num_nod=Bulk_mesh_pt->nboundary_node(ibound);
    for (unsigned inod=0;inod<num_nod;inod++)
    {
        // Get pointer to node
        Node* nod_pt=Bulk_mesh_pt->boundary_node_pt(ibound,inod);

        // Pinned in r, z and theta
        nod_pt->pin(0);nod_pt->pin(1);nod_pt->pin(2);
        nod_pt->pin(3);nod_pt->pin(4);nod_pt->pin(5);

        // Set the displacements
        nod_pt->set_value(0,0.0);
        nod_pt->set_value(1,0.0);
        nod_pt->set_value(2,0.0);
        nod_pt->set_value(3,0.0);
        nod_pt->set_value(4,0.0);
        nod_pt->set_value(5,0.0);
    }
}
// Complete the problem setup to make the elements fully functional
// Loop over the elements
unsigned n_el = Bulk_mesh_pt->nelement();
for(unsigned e=0;e<n_el;e++)
{
    // Cast to a bulk element
    ELEMENT *el_pt = dynamic_cast<ELEMENT*>(Bulk_mesh_pt->element_pt(e));
    // Set the pointer to Poisson's ratio
    el_pt->nu_pt() = &Global_Parameters::Nu;
    // Set the pointer to Fourier wavenumber
    el_pt->fourier_wavenumber_pt() = &Global_Parameters::Fourier_wavenumber;
    // Set the pointer to non-dim Young's modulus
    el_pt->youngs_modulus_pt() = &Global_Parameters::E;
    // Set the pointer to square of the angular frequency
    el_pt->omega_sq_pt() = &Global_Parameters::Omega_sq;
} // end loop over elements
// Loop over the traction elements
unsigned n_traction = Surface_mesh_pt->nelement();
for(unsigned e=0;e<n_traction;e++)
{
    // Cast to a surface element
    TimeHarmonicFourierDecomposedLinearElasticityTractionElement<ELEMENT>*
    el_pt =
    dynamic_cast<TimeHarmonicFourierDecomposedLinearElasticityTractionElement
    <ELEMENT>*>(Surface_mesh_pt->element_pt(e));

    // Set the applied traction
    el_pt->traction_fct_pt() = &Global_Parameters::boundary_traction;

} // end loop over traction elements
}
//====start_of_traction=====
// Make traction elements along the boundary r=rmin
//=====
template<class ELEMENT>
void FourierDecomposedTimeHarmonicLinearElasticityProblem<ELEMENT>::
assign_traction_elements()
{
    unsigned bound, n_neigh;
    // How many bulk elements are next to boundary 3
    bound=3;
    n_neigh = Bulk_mesh_pt->nboundary_element(bound);
    // Now loop over bulk elements and create the face elements
    for(unsigned n=0;n<n_neigh;n++)
    {
        // Create the face element
        FiniteElement *traction_element_pt
        = new TimeHarmonicFourierDecomposedLinearElasticityTractionElement<ELEMENT>
        (Bulk_mesh_pt->boundary_element_pt(bound,n),
        Bulk_mesh_pt->face_index_at_boundary(bound,n));

        // Add to mesh
        Surface_mesh_pt->add_element_pt(traction_element_pt);
    }
} // end of assign_traction_elements
//====start_of_delete_traction=====
// Delete traction elements
//=====
template<class ELEMENT>
void FourierDecomposedTimeHarmonicLinearElasticityProblem<ELEMENT>::
delete_traction_elements()
{
    // How many surface elements are in the surface mesh
    unsigned n_element = Surface_mesh_pt->nelement();

```

```

// Loop over the surface elements
for(unsigned e=0;e<n_element;e++)
{
    // Kill surface element
    delete Surface_mesh_pt->element_pt(e);
}

// Wipe the mesh
Surface_mesh_pt->flush_element_and_node_storage();
} // end of delete_traction_elements
//==start_of_doc_solution=====
/// Doc the solution
//=====
template<class ELEMENT>
void FourierDecomposedTimeHarmonicLinearElasticityProblem<ELEMENT>::
doc_solution(DocInfo& doc_info)
{
    ofstream some_file;
    char filename[100];

    // Number of plot points
    unsigned npts=5;

    // Output solution
    sprintf(filename,"%s/soln.dat",doc_info.directory().c_str());
    some_file.open(filename);
    Bulk_mesh_pt->output(some_file,npts);
    some_file.close();
    // Output norm of solution (to allow validation of solution even
    // if triangle generates a slightly different mesh)
    sprintf(filename,"%s/norm.dat",doc_info.directory().c_str());
    some_file.open(filename);
    double norm=0.0;
    unsigned nel=Bulk_mesh_pt->nelement();
    for (unsigned e=0;e<nel;e++)
    {
        double el_norm=0.0;
        Bulk_mesh_pt->compute_norm(el_norm);
        norm+=el_norm;
    }
    some_file << norm << std::endl;
} // end_of_doc_solution
//==start_of_main=====
/// Driver code
//=====
int main(int argc, char* argv[])
{
    // Number of elements in r-direction
    unsigned nr=10;

    // Number of elements in z-direction (for (approximately) square elements)
    unsigned nz=unsigned(double(nr)*Global_Parameters::Lz/Global_Parameters::Lr);
    // Set up doc info
    DocInfo doc_info;

    // Set output directory
    doc_info.set_directory("RESLT");
#ifdef ADAPTIVE
    // Set up problem
    FourierDecomposedTimeHarmonicLinearElasticityProblem
    <ProjectableTimeHarmonicFourierDecomposedLinearElasticityElement
    <TTimeHarmonicFourierDecomposedLinearElasticityElement<3> > >
    problem(nr,nz,Global_Parameters::rmin,Global_Parameters::rmax,
            Global_Parameters::zmin,Global_Parameters::zmax);
    // Solve
    unsigned max_adapt=3;
    problem.newton_solve(max_adapt);
#else
    // Set up problem
    FourierDecomposedTimeHarmonicLinearElasticityProblem
    <QTimeHarmonicFourierDecomposedLinearElasticityElement<3> >
    problem(nr,nz,Global_Parameters::rmin,Global_Parameters::rmax,
            Global_Parameters::zmin,Global_Parameters::zmax);

    // Solve
    problem.newton_solve();
#endif

    // Output the solution
    problem.doc_solution(doc_info);
} // end_of_main

```

## 1.4 Source files for this tutorial

- The source files for this tutorial are located in the directory:

```
demo_drivers/time_harmonic_fourier_decomposed_linear_↵  
elasticity/cylinder/
```

- The driver code is:

```
demo_drivers/time_harmonic_fourier_decomposed_linear_↵  
elasticity/cylinder/pressure_loaded_cylinder.cc
```

---

## 1.5 PDF file

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