

## Chapter 1

# Demo problem: Bending of a 3D non-symmetric cantilever beam made of incompressible material

In this tutorial we demonstrate the solution of a 3D solid mechanics problem: the large-amplitude bending deformation of a non-symmetric cantilever beam made of incompressible Mooney-Rivlin material.

Here is an animation of the beam's deformation. In its undeformed configuration, the beam is straight and its cross-section is given by a quarter circle. The beam is loaded by an increasing gravitational body force, acting in the negative  $y$ -direction, while its left end (at  $z = 0$ ) is held fixed. Because of its non-symmetric cross-section, the beam's downward bending deformation is accompanied by a sideways deflection.



Figure 1.1 Animation of the beam's bending deformation.

Note how the automatic mesh adaptation refines the mesh in the region of strongest bending.

## 1.1 The mesh

We use multiple inheritance to upgrade the already-existing refineable "quarter tube mesh" to a solid mesh. Following a call to the constructor of the underlying meshes, we set the nodes' Lagrangian coordinates to their current Eulerian positions to make the initial configuration stress-free.

```
//=====start_mesh=====
/// Simple quarter tube mesh upgraded to become a solid mesh
//=====
template<class ELEMENT>
class RefineableElasticQuarterTubeMesh :
public virtual RefineableQuarterTubeMesh<ELEMENT>,
public virtual SolidMesh
{

public:

    /// Constructor:
    RefineableElasticQuarterTubeMesh (GeomObject* wall_pt,
                                     const Vector<double>& xi_lo,
                                     const double& fract_mid,
                                     const Vector<double>& xi_hi,
                                     const unsigned& nlayer,
                                     TimeStepper* time_stepper_pt=
                                     &Mesh::Default_TimeStepper) :
        QuarterTubeMesh<ELEMENT>(wall_pt,xi_lo,fract_mid,xi_hi,
                                nlayer,time_stepper_pt),
        RefineableQuarterTubeMesh<ELEMENT>(wall_pt,xi_lo,fract_mid,xi_hi,
                                nlayer,time_stepper_pt)
    {
        //Assign the Lagrangian coordinates
        set_lagrangian_nodal_coordinates();
    }

    /// Empty Destructor
    virtual ~RefineableElasticQuarterTubeMesh() { }
};
```

## 1.2 Global parameters and functions

As usual, we define a namespace, `Global_Physical_Variables`, to define the problem parameters: the length of the cantilever beam,  $L$ , a (pointer to) a strain energy function, the constitutive parameters  $C_1$  and  $C_2$  for the Mooney-Rivlin strain energy function, and a (pointer to) a constitutive equation. Finally, we define the gravitational body force which acts in the negative  $y$ -direction.

```
//=====start_namespace=====
/// Global variables
//=====
namespace Global_Physical_Variables
{

    /// Length of beam
    double L=10.0;

    /// Pointer to strain energy function
    StrainEnergyFunction* Strain_energy_function_pt=0;

    /// First "Mooney Rivlin" coefficient
    double C1=1.3;

    /// Second "Mooney Rivlin" coefficient
    double C2=1.3;

    /// Pointer to constitutive law
    ConstitutiveLaw* Constitutive_law_pt=0;

    /// Non-dim gravity
    double Gravity=0.0;

    /// Non-dimensional gravity as body force
    void gravity(const double& time,
               const Vector<double> &xi,
               Vector<double> &b)
    {
        b[0]=0.0;
        b[1]=-Gravity;
        b[2]=0.0;
    }

} //end namespace
```

## 1.3 The driver code

If the code is executed without command line arguments we perform a single simulation. We start by creating the strain energy function and pass it to the constructor of the strain-energy-based constitutive equation. We then build the problem object, using `oomph-lib`'s large-displacement Taylor-Hood solid mechanics elements which are based on a continuous-pressure/displacement formulation.

```

//=====start_of_main=====
/// Driver for 3D cantilever beam loaded by gravity
//=====
int main(int argc, char* argv[])
{
    // Run main demo code if no command line arguments are specified
    if (argc==1)
    {

        // Create incompressible Mooney Rivlin strain energy function
        Global_Physical_Variables::Strain_energy_function_pt =
            new MooneyRivlin(&Global_Physical_Variables::C1,
                           &Global_Physical_Variables::C2);

        // 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 continous pressure/displacement
        CantileverProblem<RefineableQPVDElementWithContinuousPressure<3> > problem;

```

We document the initial configuration before starting a parameter study in which the magnitude of the gravitational body force is increased in small steps:

```

    // Doc solution
    problem.doc_solution();

    // Initial values for parameter values
    Global_Physical_Variables::Gravity=0.0;

    //Parameter incrementation
    unsigned nstep=10;

    double g_increment=5.0e-4;
    for(unsigned i=0;i<nstep;i++)
    {
        // Increment load
        Global_Physical_Variables::Gravity+=g_increment;

        // Solve the problem with Newton's method, allowing
        // up to max_adapt mesh adaptations after every solve.
        unsigned max_adapt=1;
        problem.newton_solve(max_adapt);

        // Doc solution
        problem.doc_solution();
    }
} // end main demo code

```

If the code is executed with a non-zero number of command line arguments, it performs a large number of additional self tests that we will not discuss here. See the driver code [three\\_d\\_cantilever.cc](#) for details.

## 1.4 The problem class

The problem class contains the usual member functions. No action is required before the mesh adaptation; we overload the function `Problem::actions_after_adapt()` to pin the redundant solid pressure degrees of freedom afterwards.

```

//=====begin_problem=====
/// Problem class for the 3D cantilever "beam" structure.
//=====
template<class ELEMENT>
class CantileverProblem : public Problem
{
public:

    /// Constructor:
    CantileverProblem();

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

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

    /// Actions before adapt. Empty

```

```

void actions_before_adapt(){}

/// Actions after adapt
void actions_after_adapt()
{
    // Pin the redundant solid pressures (if any)
    PVDEquationsBase<3>::pin_redundant_nodal_solid_pressures(
        mesh_pt()->element_pt());
}

/// Doc the solution
void doc_solution();

```

We overload the Problem::mesh\_pt() function to return a pointer to the specific mesh used in this problem:

```

/// Access function for the mesh
RefineableElasticQuarterTubeMesh<ELEMENT>* mesh_pt()
{
    return dynamic_cast<RefineableElasticQuarterTubeMesh<ELEMENT>*>(
        Problem::mesh_pt());
}

```

The private member data stores a DocInfo object in which we will store the name of the output directory.

private:

```

/// DocInfo object for output
DocInfo Doc_info;

```

```
};
```

## 1.5 The problem constructor

We start by creating the GeomObject that defines the curvilinear boundary of the beam: a circular cylinder of unit radius.

```

//=====start_of_constructor=====
/// Constructor:
//=====
template<class ELEMENT>
CantileverProblem<ELEMENT>::CantileverProblem()
{
    // Create geometric object that defines curvilinear boundary of
    // beam: Elliptical tube with half axes = radius = 1.0
    double radius=1.0;
    GeomObject* wall_pt=new EllipticalTube(radius,radius);

    // Bounding Lagrangian coordinates
    Vector<double> xi_lo(2);
    xi_lo[0]=0.0;
    xi_lo[1]=0.0;
    Vector<double> xi_hi(2);
    xi_hi[0]= Global_Physical_Variables::L;
    xi_hi[1]=2.0*atan(1.0);
}

```

We build the mesh, using six axial layers of elements, before creating an error estimator and specifying the error targets for the adaptive mesh refinement.

```

// # of layers
unsigned nlayer=6;
//Radial divider is located half-way along the circumference
double frac_mid=0.5;
//Now create the mesh
Problem::mesh_pt() = new RefineableElasticQuarterTubeMesh<ELEMENT>
    (wall_pt,xi_lo,frac_mid,xi_hi,nlayer);

// Set error estimator
dynamic_cast<RefineableElasticQuarterTubeMesh<ELEMENT>*>(
    mesh_pt())->spatial_error_estimator_pt()=new ZZErrorEstimator;
// Error targets for adaptive refinement
mesh_pt()->max_permitted_error()=0.05;
mesh_pt()->min_permitted_error()=0.005;

```

We complete the build of the elements by specifying the constitutive equation and the body force. We check that the element is based on a pressure/displacement formulation, and, if so, select an incompressible formulation. (This check is only required because the self-tests not shown here also include cases in which the problem is solved using a displacement-based formulation with compressible elasticity; see also the section [How to enforce incompressibility](#) below).

```

// Complete build of elements
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::gravity;
// Material is incompressible: Use incompressible displacement/pressure
// formulation (if the element is pressure based, that is!)
PVDEquationsWithPressure<3>* cast_el_pt =
    dynamic_cast<PVDEquationsWithPressure<3>>(mesh_pt()->element_pt(i));
if (cast_el_pt!=0)
{
    cast_el_pt->set_incompressible();
}
} // done build of elements

```

We fix the position of all nodes at the left end of the beam (on boundary 0) and pin any redundant solid pressures.

```

// Pin the left boundary (boundary 0) in all directions
unsigned b=0;
unsigned n_side = mesh_pt()->nboundary_node(b);

//Loop over the nodes
for(unsigned i=0;i<n_side;i++)
{
    mesh_pt()->boundary_node_pt(b,i)->pin_position(0);
    mesh_pt()->boundary_node_pt(b,i)->pin_position(1);
    mesh_pt()->boundary_node_pt(b,i)->pin_position(2);
}
// Pin the redundant solid pressures (if any)
PVDEquationsBase<3>::pin_redundant_nodal_solid_pressures(
    mesh_pt()->element_pt(i));

```

Finally, we assign the equation numbers and define the output directory.

```

//Assign equation numbers
assign_eqn_numbers();
// Prepare output directory
Doc_info.set_directory("RESULT");

} //end of constructor

```

## 1.6 Post-processing

The post-processing function `doc_solution()` simply outputs the shape of the deformed beam.

```

//=====start_doc=====
/// Doc the solution
//=====
template<class ELEMENT>
void CantileverProblem<ELEMENT>::doc_solution()
{
    ofstream some_file;
    char filename[100];
    // Number of plot points
    unsigned n_plot = 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,n_plot);
    some_file.close();
    // Increment label for output files
    Doc_info.number()++;
} //end doc

```

## 1.7 Comments and exercises

### 1.7.1 How to enforce incompressibility

We stress that the imposition of incompressibility must be requested explicitly via the element's member function `incompressible()`. Mathematically, incompressibility is enforced via a Lagrange multiplier which manifests itself physically as the pressure. Incompressibility can therefore only be enforced for elements that employ the pressure-displacement formulation of the principle of virtual displacements. This is why we the `Problem` constructor checked if the element is derived from the `PVDEquationsWithPressure` class before setting the element's `incompressible` flag to true.

As usual, `oomph-lib` provides self-tests that assess if the enforcement incompressibility (or the lack thereof) is consistent:

- The compiler will not allow the user to enforce incompressibility on elements that are based on the displacement form of the principle of virtual displacements.

- Certain constitutive laws, such as the Mooney-Rivlin law used in the present example require an incompressible formulation. If `oomph-lib` is compiled with the `PARANOID` flag, an error is thrown if such a constitutive law is used by an element for which incompressibility has not been requested.

**Recall that the default setting is not to enforce incompressibility!**

If the library is compiled without the `PARANOID` flag no warning will be issued but the results will be "wrong" at least in the sense that the material does not behave like an incompressible Mooney-Rivlin solid. In fact, it is likely that the Newton solver will diverge. Anyway, as we keep saying, without the `PARANOID` flag, you're on your own!

- Some constitutive laws can be used for compressible and incompressible behaviour. In this case it is important to set the `incompressible()` flag to the correct value. This issue is discussed in more detail in [another tutorial](#).

You should experiment with different combinations of constitutive laws and element types to familiarise yourself with [these](#).

---

## 1.8 Source files for this tutorial

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

`demo_drivers/solid/three_d_cantilever/`

- The driver code is:

`demo_drivers/solid/three_d_cantilever/three_d_cantilever.cc`

---

## 1.9 PDF file

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