Created in COMSOL Multiphysics 5.4

Loaded Spring — Using Global Equations to Satisfy Constraints

Introduction

In this tutorial example, which demonstrates a more generally applicable method, a structural mechanics model of a spring is augmented by a global equation that solves for the load required to achieve a desired total extension of the spring.

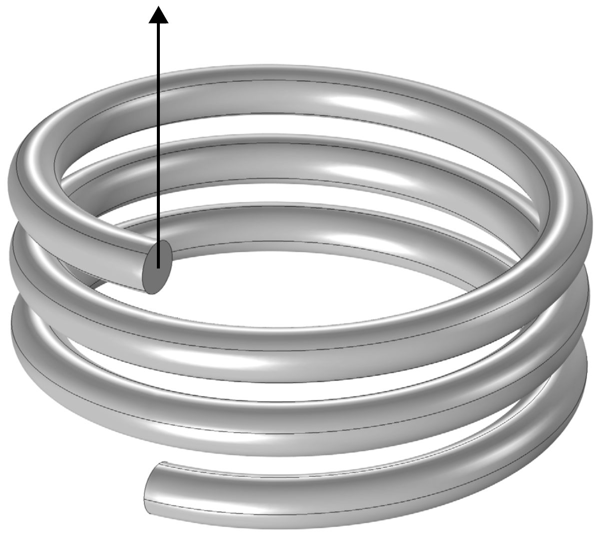


Figure 1: A three-turn steel spring is fixed at one end, and has a load applied at the other. The load is a variable which is solved for to achieve a total displacement.

Model Definition

Figure 1 shows the modeled three-turn steel spring. One end of the spring is fixed rigidly, and the other end has a distributed load applied to it, acting in the axial direction of the spring. Rather than an input to the model, this load is a variable being solved for; it is implicitly specified via a global equation in such a way as to give a total spring extension of 2 cm. The extension of the spring is computed by using an average operator on the moving end of the spring. The average operator evaluates the average z-displacement over the boundary at which the load is applied.

The global equation adds one additional degree of freedom to the model, the unknown load. Not all available equations solvers are suited for such problems, but the direct solver used as default for structural mechanics can handle it. Because the structure has a uniform cross section, use a swept mesh.

Results and Discussion

Figure 2 shows the deformed shape of the spring. The average displacement of the end of the spring is 2 cm, as specified by the global equation. The force required to get this displacement is 705 N. Although this problem uses a linear elastic material model, this approach would work equally well if the material model was nonlinear or if geometric nonlinearity was taken into account.

Global equations do have certain restrictions upon their usage. The global equation must be continuous and differentiable with respect to all of the unknowns, and it must not overconstrain, nor underconstrain, the problem. Each global equation should add one constraint and one degree of freedom to the model. Under these conditions, the global equations can be used in a variety of ways beyond what is shown here.

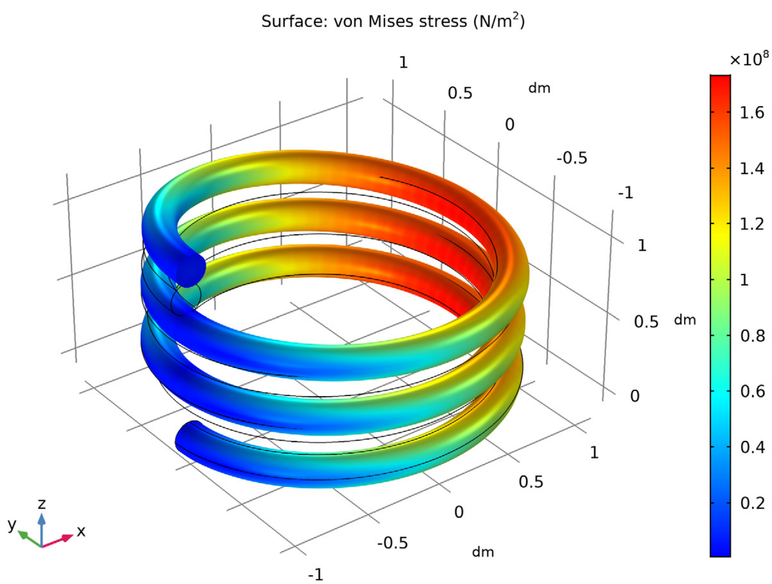


Figure 2: The deformed shape of the spring.

Application Library path: COMSOL_Multiphysics/Structural_Mechanics/loaded_spring

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
dh	2[cm]	0.02 m	Prescribed extension

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **dm**.

Helix 1 (hel1)

Create a helix for the spring (Figure 1).

1 In the **Geometry** toolbar, click **Helix**.

2 In the **Settings** window for **Helix**, locate the **Rotation Angle** section.

3 In the **Rotation** text field, type `180`.

4 Click **Build All Objects**.

DEFINITIONS

Next, add an **Average** operator that you will later use to average the *z*-directional displacement field on the end of the spring.

Average 1 (aveop1)

1 In the **Definitions** toolbar, click **Component Couplings** and choose **Average**.

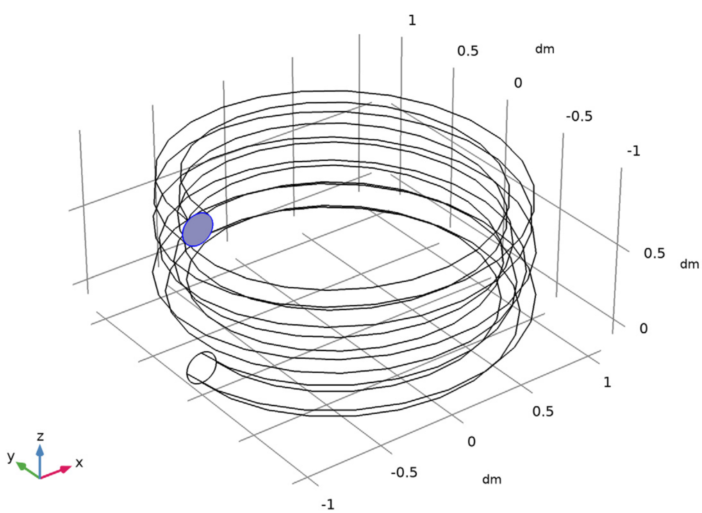
Choose wireframe rendering to get a better view on some boundaries where you will assign boundary conditions.

2 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

3 In the **Settings** window for **Average**, locate the **Source Selection** section.

4 From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundary 4 only.



SOLID MECHANICS (SOLID)

Next, set up the physics. Add a global equation to compute the appropriate load for the prescribed extension. As an advanced feature, the **Global Equations** entry is not available by default in the context menu.

1 In the **Model Builder** window's toolbar, click the **Show** button and select **Advanced Physics Options** in the menu.

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)
`Force`	`aveop1(w)-dh`	0	0

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `force` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Force (N)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `displacement` in the text field.

12 Click **Filter**.

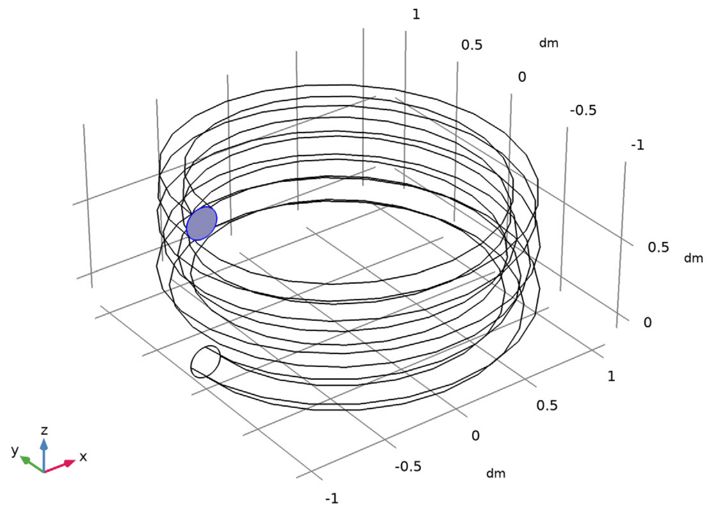
13 In the tree, select **General>Displacement (m)**.

14 Click **OK**.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 4 only.



3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

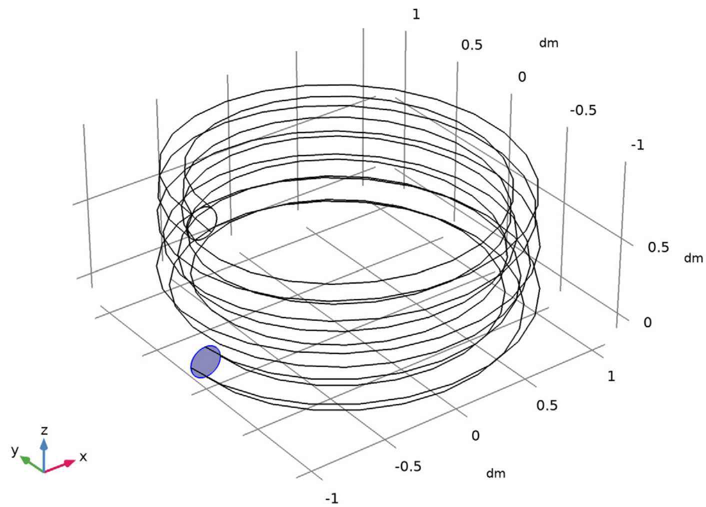
5 Specify the $\mathbf{F}_{tot}$ vector as

0	x
0	y
Force	z

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 3 only.



MATERIALS

Assign material properties. Use **Steel AISI 4340** for all domains.

ADD MATERIAL

1. In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **Built-In>Steel AISI 4340**.
4. Click **Add to Component** in the window toolbar.
5. In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

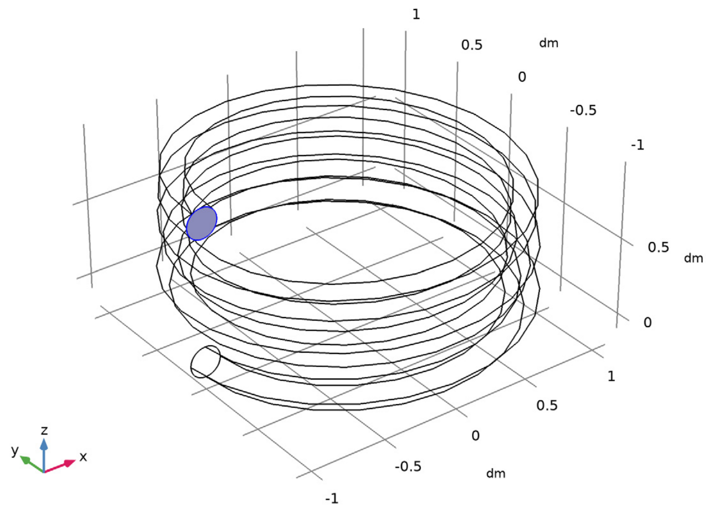
MESH 1

Use swept mesh to generate a uniform mesh over the spring domain. Start by specifying the mesh on one end face of the spring.

Free Triangular 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundary 4 only.



Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Coarser**.

Distribution 1

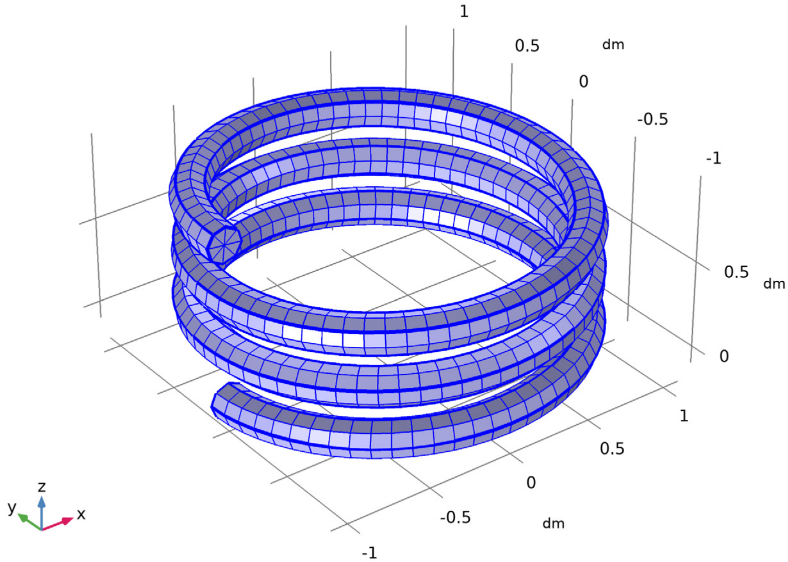
1 In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

2 Right-click **Swept 1** and choose **Distribution**.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 200.

5 Click **Build All**.



STUDY 1

In the **Home** toolbar, click **Compute**.

RESULTS

Stress (solid)

The default plot shows the von Mises stress on the surface of the spring. Compare the plot with Figure 2.

Derived Values

Evaluate the force required to get the displacement specified in the global equations.

Global Evaluation 1

1. In the **Results** toolbar, click **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
3. In the table, enter the following settings:

Expression	Unit	Description
Force	N	State variable Force

4 Click **Evaluate**.

Derived Values

Finish the result analysis by evaluating the average displacement of the end of the spring.

Global Evaluation 2

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`aveop1(w)`	`dm`	`Average 1`

4 Click **Evaluate**.