Model created in COMSOL Multiphysics 6.4

Uniaxial Stretching of a Rectangular Sheet

This model is licensed under the COMSOL Software License Agreement 6.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

Wrinkling is a commonly occurring phenomenon in thin sheets, caused by either loading or manufacturing defects. The numerical treatment of wrinkles in thin structures can be approached using the following two theories:

- *Membrane theory along with tension field theory* — Membrane theory models wrinkles as a mean surface and gives a correct stress distribution in the wrinkled region. This is computationally inexpensive compared to shell theory, but the wrinkle amplitudes and wavelengths cannot be obtained. An example of modeling wrinkling in a rectangular sheet using the Membrane interface can be found in the Structural Mechanics Module Application Library model *Uniaxial Stretching of a Rectangular Membrane*.
- *Shell theory* — As wrinkling is a local buckling phenomenon, nonlinear postbuckling analysis is needed to find the amplitudes and wavelengths of the wrinkles. This modeling approach is nontrivial and computationally expensive.

In this example, wrinkling in a thin rectangular sheet is modeled using a shell theory. First, a static analysis is performed to determine the region of negative principal stresses without assuming wrinkles. The negative principal stress is a prerequisite for the occurrence of wrinkles. Next, a prestressed buckling analysis is carried out to find out the potential linearized buckling modes. Finally, a nonlinear postbuckling analysis is carried out to investigate the evolution of wrinkles. The example is taken from Ref. 1, with which the results obtained here are compared.

Model Definition

A rectangular sheet 25 cm in length, 10 cm in width, and 0.1 mm thick is stretched in the longitudinal direction.

One of the short edges is fixed, while a prescribed displacement is applied on the opposite edge. The long edges are unconstrained.

The sheet is made of an incompressible neo-Hookean hyperelastic material with a Lame parameter of 6 MPa.

Results and Discussions

The example is based on Ref. 1 and Ref. 2 with some modifications. The authors (see Ref. 2) used 100,000 quadrilateral mesh elements, whereas we use 12,500 elements to save computation time. The nominal strain at which wrinkling starts in the postbuckling analysis depends on the mesh distribution, as reported in Ref. 2 (although the variation is small). Also the buckling modes and the critical factor depend on the mesh distribution. The authors in Ref. 1 reported that the numerical results of the postbuckling analysis depends on the number of modes used as geometric imperfection, as well as on their amplitudes. They have considered four buckling modes and an amplitude equal to 0.1% of the sheet thickness. In this example, four buckling modes are also used but the amplitude varies for each mode. A lower factor is assigned to the higher modes to reduce their influence.

Figure 1 shows the regions of negative transverse stress, called wrinkled regions, for different values of the nominal strain. A static analysis is carried out without any assumption of wrinkles. The occurrence of negative transverse stress shows the possibility of wrinkling. The size of the wrinkled region is reduced with increasing nominal strain. Figure 2 and Figure 3 show the variation of the transverse stress along a central longitudinal and a transverse line, respectively, for different nominal strains. The results are similar to those presented in Ref. 1.

Figure 4 through Figure 7 show the different wrinkling or buckling modes computed in the prestressed buckling analysis. The modes are computed using a nominal strain of 1%, which is the same as in Ref. 1.

Figure 8 shows the wrinkles produced in the postbuckling analysis with different nominal strains. It can be seen that the wrinkle amplitude increases initially and then decreases, which is consistent with results presented in Ref. 1. This behavior becomes clearer in Figure 9, where the wrinkle amplitude is plotted along the transverse line for different nominal strains. The results are consistent with those in the corresponding figure in Ref. 1. The wrinkle amplitude is at a maximum at the center due to the symmetries of the model. At the nominal strain 30%, the wrinkled amplitude becomes very small. As in Ref. 1, when the wrinkle amplitude is plotted against the nominal strain (see Figure 10), a bell-shaped curve is observed. The amplitude values are close to those of Ref. 1. Some small differences are observed due to the modeling differences discussed above.

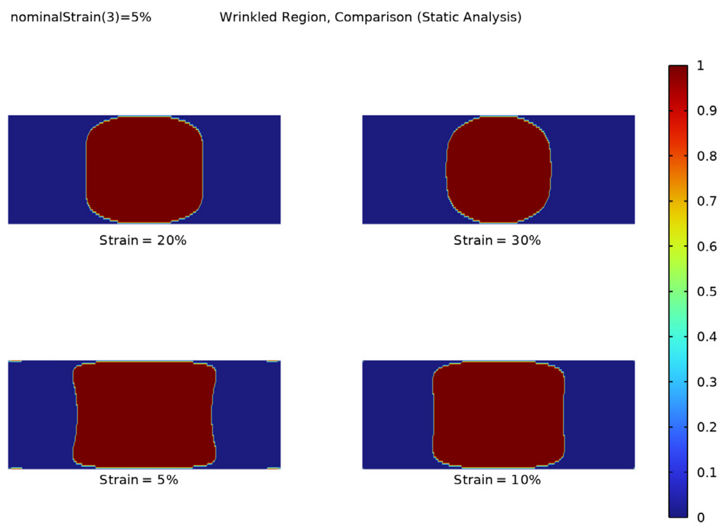

Figure 1: Wrinkled region in the static analysis.

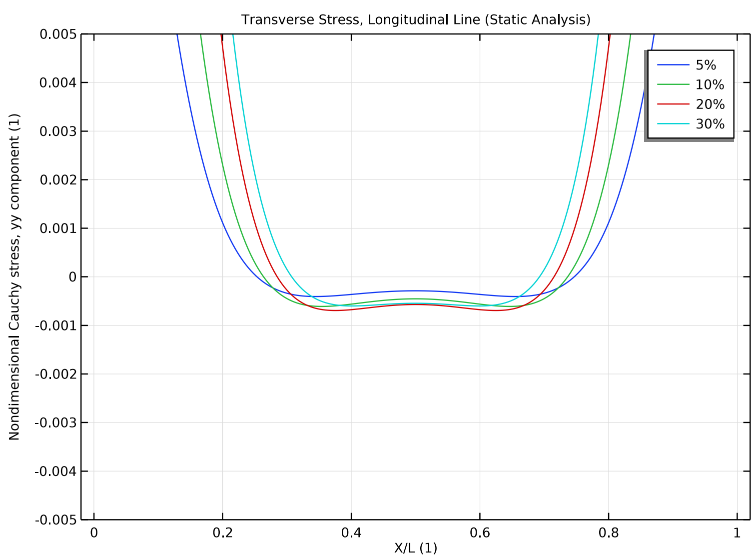

Figure 2: Transverse stress along the longitudinal line in the static analysis.

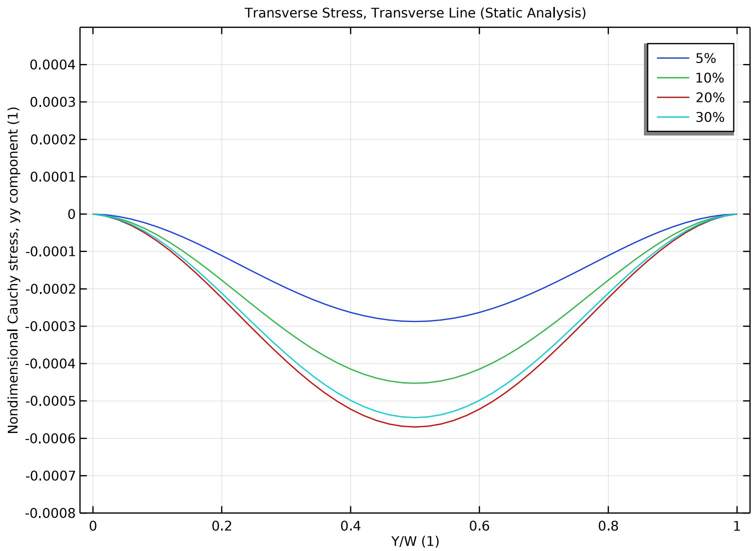

Figure 3: Transverse stress along the transverse line in the static analysis.

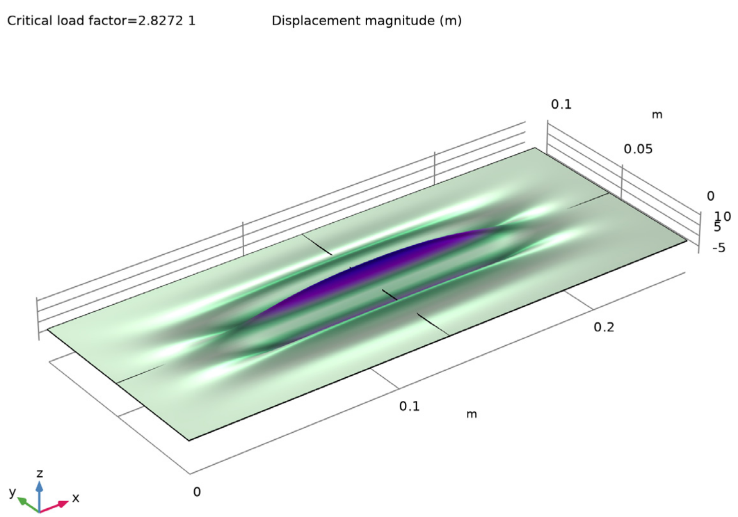

Figure 4: First mode shape in the prestressed buckling analysis.

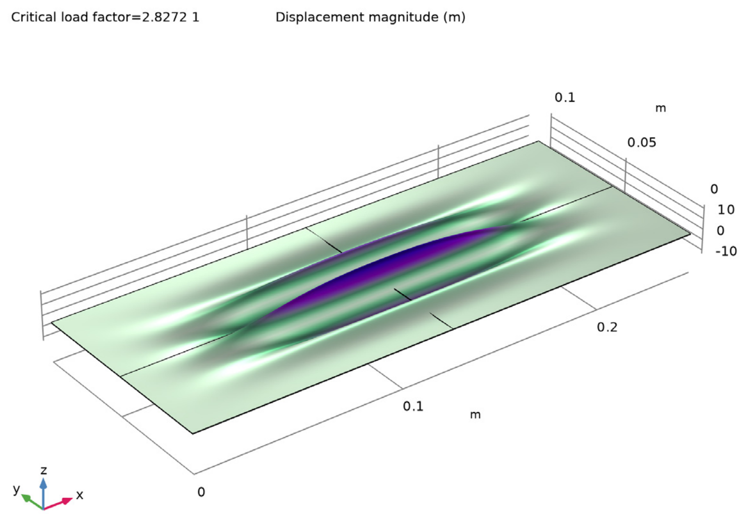

Figure 5: Second mode shape in the prestressed buckling analysis.

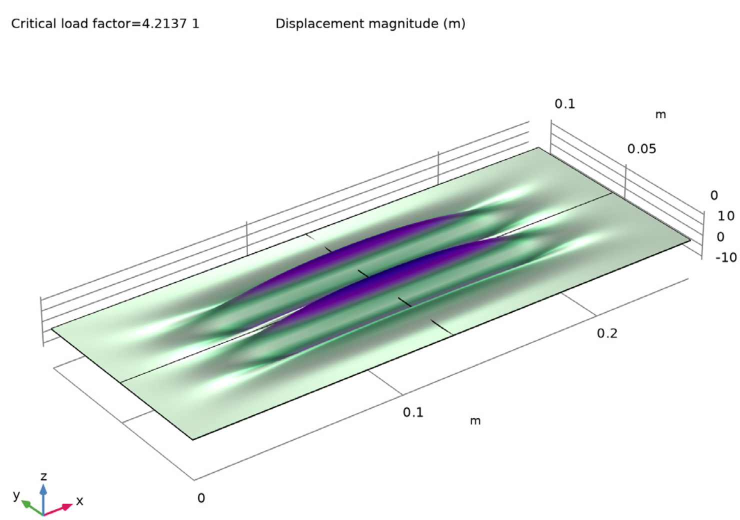

Figure 6: Third mode shape in the prestressed buckling analysis.

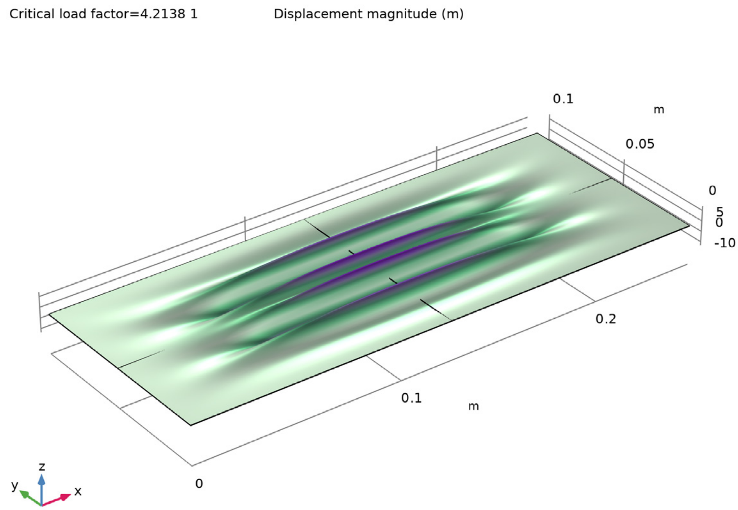

Figure 7: Fourth mode shape in the prestressed buckling analysis.

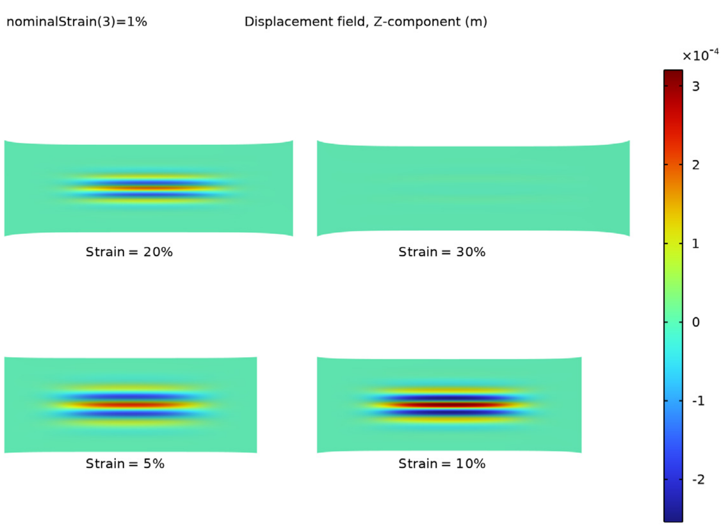

Figure 8: Out of plane displacement in the postbuckling analysis.

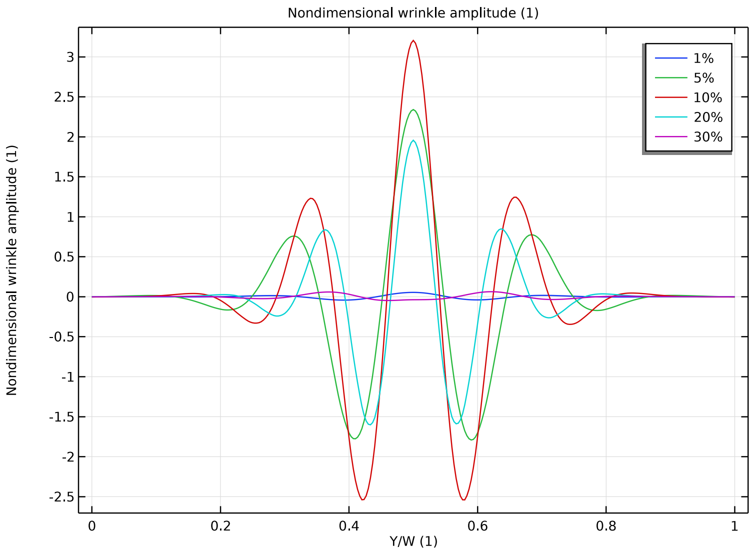

Figure 9: Wrinkle amplitude in the postbuckling analysis.

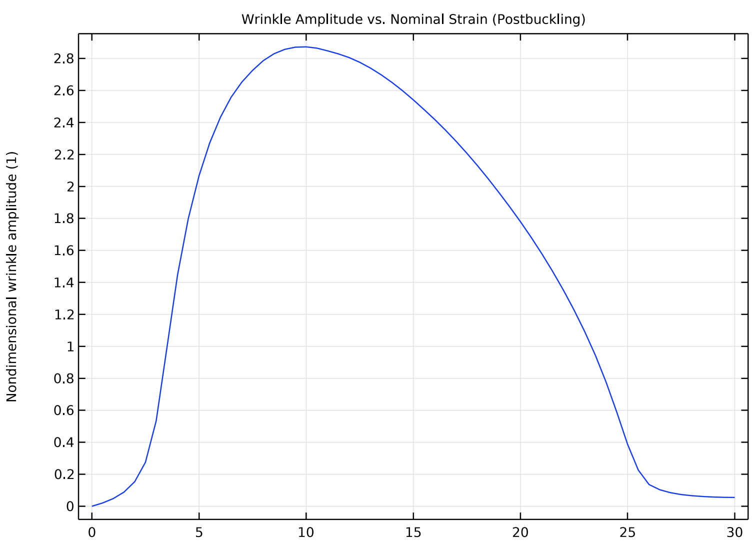

Figure 10: Wrinkle amplitude versus nominal strain in the postbuckling analysis.

Notes About the COMSOL Implementation

The sheet will easily buckle under compressive loads, so modes with a small negative critical load factors are not the desired ones. To find the actual wrinkling modes using the **Linear Buckling** study, the settings in the **Eigenvalue Solver** node must be tuned. The results in Ref. 1 and Ref. 2 show that wrinkles appear in the sheet at nominal strains around 1% to 5%. The sheet has a prestretch of 1%, which means that the critical load factor is around 1 for the wrinkling modes. Hence, in the **Eigenvalue Solver** node, enter 1 in the text field **Search for eigenvalues around shift**. Also, set the **Search method around shift** to **Larger real part**.

The settings for the postbuckling analysis are made easier by the **Buckling Imperfection** node. It contains a **Deformed Geometry** section that allows choosing the buckling solution and modes that will define the prescribed displacements. The **Configure** button generates one **Prescribed Deformation** node for each structural mechanics physics interface involved, and sets the prescribed deformations with the variables defined by the node. The prescribed deformations are those computed in the linear buckling study multiplied by the scale factor. Low values of the scale factor make the buckling effect sharper, at the expense of the convergence properties.

References

1. V. Nayyar, K. Ravi-Chandar, and R Huang, "Stretch-induced stress patterns and wrinkles in hyperelastic thin sheets," *Int. J. Solids Struct.*, vol. 48, pp. 3471–3483, 2011.

2. V. Nayyar, "Stretch-induced compressive stress and wrinkling in elastic thin sheets," Master's Thesis, The University of Texas at Austin, Austin, TX, 2010.

Application Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/sheet_uniaxial_stretching`

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 > Shell (shell)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies > Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Geometric Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, type `Geometric Parameters` in the **Label** text field.
3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
mu	6[MPa]	6E6 Pa	Lame parameter
W	10[cm]	0.1 m	Width of sheet
alpha	2.5	2.5	Aspect ratio of sheet
L	alpha*W	0.25 m	Length of sheet
th	W/1000	1E-4 m	Thickness of sheet
numX	L/1[mm]	250	Number of mesh elements in X direction
numY	W/2[mm]	50	Number of mesh elements in Y direction
nominalStrain	1[%]	0.01	Nominal strain
geomImpFactor	1E4	10000	Geometric imperfection factor

GEOMETRY 1

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1) > Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `L`.

4 In the **Height** text field, type `W`.

5 Click **Build Selected**.

Add longitudinal and transverse central lines for result evaluation.

Work Plane 1 (wp1) > Line Segment 1 (ls1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **yw** text field, type `0.5*W`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **xw** text field, type `L`.

7 In the **yw** text field, type `0.5*W`.

Work Plane 1 (wp1) > Line Segment 2 (ls2)

1 Right-click **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** > **Plane Geometry** > **Line Segment 1 (ls1)** and choose **Duplicate**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 In the **xw** text field, type `0.5*L`.

4 In the **yw** text field, type `0`.

5 Locate the **Endpoint** section. In the **xw** text field, type `0.5*L`.

6 In the **yw** text field, type `W`.

7 Click **Build Selected**.

DEFINITIONS

Maximum 1 (maxop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.

2 In the **Settings** window for **Maximum**, locate the **Source Selection** section.

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

4 From the **Selection** list, choose **All boundaries**.

Minimum 1 (minop1)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Minimum**.
2. In the **Settings** window for **Minimum**, locate the **Source Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **All boundaries**.

SHELL (SHELL)

Hyperelastic Material, Layered 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Hyperelastic Material, Layered**.
2. Select Boundary 1 only.
3. In the **Settings** window for **Hyperelastic Material, Layered**, locate the **Boundary Selection** section.
4. From the **Selection** list, choose **All boundaries**.
5. Locate the **Hyperelastic Material** section. From the **Compressibility** list, choose **Incompressible**.
6. From the **Use mixed formulation** list, choose **Implicit formulation**.

Fixed Constraint 1

1. In the **Physics** toolbar, click **Edges** and choose **Fixed Constraint**.
2. Select Edges 1 and 3 only.

Prescribed Displacement/Rotation 1

1. In the **Physics** toolbar, click **Edges** and choose **Prescribed Displacement/Rotation**.
2. Select Edges 11 and 12 only.
3. In the **Settings** window for **Prescribed Displacement/Rotation**, locate the **Prescribed Displacement** section.
4. From the **Displacement in x direction** list, choose **Prescribed**.
5. In the u_{0x} text field, type `nominalStrain*L`.
6. From the **Displacement in y direction** list, choose **Prescribed**.
7. From the **Displacement in z direction** list, choose **Prescribed**.

MATERIALS

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Lamé parameter μ	muLame	mu	N/m^2	Lamé parameters
Density	rho	500	kg/m^3	Basic
Thickness	lth	th	m	Shell
Mesh elements	lne	1	l	Shell

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 1 and 3 only.

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

4 In the **Number of elements** text field, type `numY/2`.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 2 and 7 only.

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

4 In the **Number of elements** text field, type `numX/2`.

5 Click **Build All**.

STUDY: STATIC ANALYSIS

1 In the **Model Builder** window, click **Study 1**.

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

3 Clear the **Generate default plots** checkbox.

4 In the **Label** text field, type `Study: Static Analysis`.

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Static Analysis** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
nominalStrain (Nominal strain)	`range(0,2.5,30)`	%

6 In the **Study** toolbar, click **Compute**.

RESULTS

Layered Material 1

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results** > **Datasets** and choose **More Datasets** > **Layered Material**.

Wrinkled Region, Comparison (Static Analysis)

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Wrinkled Region, Comparison (Static Analysis)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 1**.

4 From the **Parameter value (nominalStrain (%))** list, choose **5**.

5 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

6 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

7 Click to expand the **Plot Array** section. From the **Array type** list, choose **Square**.

8 In the **Relative row padding** text field, type `0.5`.

Surface 1

1 Right-click **Wrinkled Region, Comparison (Static Analysis)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `shell.syy<=0`.

4 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

5 From the **Resolution** list, choose **No refinement**.

6 From the **Smoothing** list, choose **None**.

Solution Array 1

1 Right-click **Surface 1** and choose **Solution Array**.

2 In the **Settings** window for **Solution Array**, locate the **Data** section.

3 From the **Parameter selection (nominalStrain)** list, choose **From list**.

4 In the **Parameter values (nominalStrain (%))** list, choose **5**, **10**, **20**, and **30**.

5 Locate the **Plot Array** section. From the **Array shape** list, choose **Square**.

Wrinkled Region, Comparison (Static Analysis)

In the **Model Builder** window, under **Results** click **Wrinkled Region, Comparison (Static Analysis)**.

Table Annotation 1

1 In the **Wrinkled Region, Comparison (Static Analysis)** toolbar, click **More Plots** and choose **Table Annotation**.

2 In the **Settings** window for **Table Annotation**, locate the **Data** section.

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	z-coordinate	Annotation
0.3*L	0	0	Strain = 5%
1.6*L	0	0	Strain = 10%
0.3*L	2.2*W	0	Strain = 20%
1.6*L	2.2*W	0	Strain = 30%

5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

Wrinkled Region, Comparison (Static Analysis)

1 Click the **Zoom Extents** button in the **Graphics** toolbar.

2 In the **Model Builder** window, click **Wrinkled Region, Comparison (Static Analysis)**.

3 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

4 From the **View** list, choose **New view**.

5 In the **Wrinkled Region, Comparison (Static Analysis)** toolbar, click **Plot**.

6 Click the **Go to XY View** button in the **Graphics** toolbar.

7 Click the **Show Grid** button in the **Graphics** toolbar.

8 Click the **Show Axis Orientation** button in the **Graphics** toolbar.

Transverse Stress, Longitudinal Line (Static Analysis)

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Transverse Stress, Longitudinal Line (Static Analysis)` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (nominalStrain)** list, choose **Manual**.

4 In the **Parameter indices (1-13)** text field, type `3,5,9,13`.

5 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

6 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

7 In the **x minimum** text field, type `-0.02`.

8 In the **x maximum** text field, type `1.02`.

9 In the **y minimum** text field, type `-0.005`.

10 In the **y maximum** text field, type `0.005`.

Line Graph 1

1 Right-click **Transverse Stress, Longitudinal Line (Static Analysis)** and choose **Line Graph**.

2 Select Edges 4 and 9 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `gpeval(4,shell.atxd1(0,mean(shell.syy)))/shell.Eequ`.

5 Select the **Description** checkbox. In the associated text field, type `Nondimensional Cauchy stress, yy component`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `X/L`.

8 Click to expand the **Legends** section. Select the **Show legends** checkbox.

9 In the **Transverse Stress, Longitudinal Line (Static Analysis)** toolbar, click **Plot**.

Transverse Stress, Transverse Line (Static Analysis)

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Transverse Stress, Transverse Line (Static Analysis)` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (nominalStrain)** list, choose **Manual**.

4 In the **Parameter indices (1-13)** text field, type `3,5,9,13`.

5 Locate the **Title** section. From the **Title type** list, choose **Label**.

6 Locate the **Axis** section. Select the **Manual axis limits** checkbox.

7 In the **x minimum** text field, type `-0.02`.

8 In the **x maximum** text field, type `1.02`.

9 In the **y minimum** text field, type `-0.0008`.

10 In the **y maximum** text field, type `0.0005`.

Line Graph 1

1 Right-click **Transverse Stress, Transverse Line (Static Analysis)** and choose **Line Graph**.

2 Select Edges 6 and 8 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `gpeval(4,shell.atxd1(0,mean(shell.syy)))/ shell.Eequ`.

5 Select the **Description** checkbox. In the associated text field, type `Nondimensional Cauchy stress, yy component`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `Y/W`.

8 Locate the **Legends** section. Select the **Show legends** checkbox.

9 In the **Transverse Stress, Transverse Line (Static Analysis)** toolbar, click **Plot**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces > Linear Buckling**.

4 Right-click and choose **Add Study**.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY: PRESTRESSED BUCKLING ANALYSIS

In the **Settings** window for **Study**, type `Study: Prestressed Buckling Analysis` in the **Label** text field.

Solution 2 (sol2)

In the **Study** toolbar, click **Show Default Solver**.

Step 2: Linear Buckling

1 In the **Model Builder** window, under **Study: Prestressed Buckling Analysis** click **Step 2: Linear Buckling**.

2 In the **Settings** window for **Linear Buckling**, locate the **Study Settings** section.

3 In the **Desired number of buckling modes** text field, type `10`.

4 Locate the **Values of Linearization Point** section. From the **Settings** list, choose **User controlled**.

5 From the **Solution** list, choose **Solution 2 (sol2)**.

6 From the **Use** list, choose **Solution Store 1 (sol3)**.

Solution 2 (sol2)

1 In the **Model Builder** window, expand the **Study: Prestressed Buckling Analysis** > **Solver Configurations** > **Solution 2 (sol2)** node.

2 In the **Model Builder** window, under **Study: Prestressed Buckling Analysis** > **Solver Configurations** > **Solution 2 (sol2)** click **Eigenvalue Solver 1**.

3 In the **Settings** window for **Eigenvalue Solver**, locate the **General** section.

4 In the **Search for eigenfrequencies around shift** text field, type `1`.

5 From the **Search method around shift** list, choose **Larger real part**.

6 In the **Study** toolbar, click **Compute**.

RESULTS

Mode Shape (Prestressed Buckling Analysis)

1 In the **Settings** window for **3D Plot Group**, type `Mode Shape (Prestressed Buckling Analysis)` in the **Label** text field.

2 In the **Mode Shape (Prestressed Buckling Analysis)** toolbar, click **Plot**.

Now, prescribe a deformation to the geometry from the calculated buckling modes to perform a postbuckling analysis.

DEFINITIONS (COMP1)

Buckling Imperfection 1 (bcki1)

1 In the **Definitions** toolbar, click **Physics Utilities** and choose **Buckling Imperfection**.

2 In the **Settings** window for **Buckling Imperfection**, locate the **Deformed Geometry** section.

3 Find the **Mode selection** subsection. In the table, enter the following settings:

Mode	Scale factor
1	`geomImpFactor`
2	`geomImpFactor/5`

Mode	Scale factor
3	geomImpFactor/10
4	geomImpFactor/20

4 Click **Configure** in the upper-right corner of the **Deformed Geometry** section. This creates a **Prescribed Deformation** node with the requested deformation settings. The newly created **Prescribed Deformation** node is automatically disabled in the existing study steps to enable further computations without changes in the results.

5 Locate the **Nonlinear Buckling Study** section. From the **Load parameter** list, choose **nominalStrain**.

6 From the **Parameter value list** list, choose **User defined**.

7 In the **Range** text field, type `range(0,0.5,30)`.

8 In the **Unit** text field, type `%`.

9 Click **Configure** in the upper-right corner of the **Nonlinear Buckling Study** section.

This button creates a new study with stationary step, activates geometric nonlinearities and applies an auxiliary sweep for the postbuckling study.

STUDY: POSTBUCKLING ANALYSIS

1 In the **Model Builder** window, click **Study 3**.

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

3 Clear the **Generate default plots** checkbox.

4 In the **Label** text field, type `Study: Postbuckling Analysis`.

5 In the **Study** toolbar, click **Compute**.

RESULTS

Layered Material 2

1 In the **Results** toolbar, click **More Datasets** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, locate the **Data** section.

3 From the **Dataset** list, choose **Study: Postbuckling Analysis/Solution 4 (sol4)**.

Out-of-Plane Displacement, Comparison (Postbuckling)

1 In the **Model Builder** window, right-click **Wrinkled Region, Comparison (Static Analysis)** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Out-of-Plane Displacement, Comparison (Postbuckling)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Layered Material 2**.

4 Locate the **Title** section. From the **Title type** list, choose **Automatic**.

Surface 1

1 In the **Model Builder** window, expand the **Out-of-Plane Displacement, Comparison (Postbuckling)** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `w`.

4 Locate the **Quality** section. From the **Resolution** list, choose **Normal**.

5 From the **Smoothing** list, choose **Inside material domains**.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Expression** section.

3 In the **z-component** text field, type `w*10`.

4 Locate the **Scale** section.

5 Select the **Scale factor** checkbox. In the associated text field, type `1`.

Solution Array 1

1 In the **Model Builder** window, click **Solution Array 1**.

2 In the **Settings** window for **Solution Array**, locate the **Data** section.

3 In the **Parameter values (nominalStrain (%))** list, choose **5**, **10**, **20**, and **30**.

Out-of-Plane Displacement, Comparison (Postbuckling)

1 In the **Model Builder** window, under **Results** click **Out-of-Plane Displacement, Comparison (Postbuckling)**.

2 In the **Out-of-Plane Displacement, Comparison (Postbuckling)** toolbar, click **Plot**.

Out-of-Plane Displacement, Comparison (Postbuckling) 1

Right-click **Out-of-Plane Displacement, Comparison (Postbuckling)** and choose **Duplicate**.

Surface 1

1 In the **Model Builder** window, expand the **Out-of-Plane Displacement, Comparison (Postbuckling) 1** node.

2 In the **Model Builder** window, expand the **Surface 1** node.

Solution Array 1, Table Annotation 1

1 In the **Model Builder** window, under **Results** > **Out-of-Plane Displacement, Comparison (Postbuckling) 1**, Ctrl-click to select **Surface 1** > **Solution Array 1** and **Table Annotation 1**.

2 Right-click and choose **Delete**.

Out-of-Plane Displacement (Postbuckling)

1 In the **Model Builder** window, under **Results** click **Out-of-Plane Displacement, Comparison (Postbuckling) 1**.

2 In the **Settings** window for **3D Plot Group**, type `Out-of-Plane Displacement (Postbuckling)` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **View** list, choose **New view**.

4 In the **Out-of-Plane Displacement (Postbuckling)** toolbar, click **Plot**.

5 Click the **Go to XY View** button in the **Graphics** toolbar.

6 Click the **Show Axis Orientation** button in the **Graphics** toolbar.

7 Click the **Show Grid** button in the **Graphics** toolbar.

Animation: Out-of-Plane Displacement (Postbuckling)

1 In the **Results** toolbar, click **Animation** and choose **Player**.

2 In the **Settings** window for **Animation**, type `Animation: Out-of-Plane Displacement (Postbuckling)` in the **Label** text field.

3 Locate the **Scene** section. From the **Subject** list, choose **Out-of-Plane Displacement (Postbuckling)**.

4 Locate the **Playing** section. In the **Display each frame for** text field, type `0.2`.

5 Click **Show Frame**.

Wrinkle Amplitude (Postbuckling)

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Wrinkle Amplitude (Postbuckling)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Postbuckling Analysis/ Solution 4 (sol4)**.

4 From the **Parameter selection (nominalStrain)** list, choose **Manual**.

5 In the **Parameter indices (1-61)** text field, type `3,11,21,41,61`.

Line Graph 1

1 Right-click **Wrinkle Amplitude (Postbuckling)** and choose **Line Graph**.

2 Select Edges 6 and 8 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `w/th`.

5 Select the **Description** checkbox. In the associated text field, type `Nondimensional wrinkle amplitude`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
7 In the **Expression** text field, type `Y/W`.
8 Locate the **Legends** section. Select the **Show legends** checkbox.
9 In the **Wrinkle Amplitude (Postbuckling)** toolbar, click **Plot**.

Wrinkle Amplitude vs. Nominal Strain (Postbuckling)

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Wrinkle Amplitude vs. Nominal Strain (Postbuckling)` in the **Label** text field.
3 Locate the **Title** section. From the **Title type** list, choose **Label**.
4 Locate the **Data** section. From the **Dataset** list, choose **Study: Postbuckling Analysis/ Solution 4 (sol4)**.

Point Graph 1

1 Right-click **Wrinkle Amplitude vs. Nominal Strain (Postbuckling)** and choose **Point Graph**.
2 Select Point 5 only.
3 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
4 In the **Expression** text field, type `0.5*(maxop1(w)-minop1(w))/th`.
5 Select the **Description** checkbox. In the associated text field, type `Nondimensional wrinkle amplitude`.
6 In the **Wrinkle Amplitude vs. Nominal Strain (Postbuckling)** toolbar, click **Plot**.

STUDY: STATIC ANALYSIS

Step 1: Stationary

In **Study 1: Static Analysis**, disable the **Prescribed Deformation** node.

1 In the **Model Builder** window, under **Study: Static Analysis** click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 Select the **Modify model configuration for study step** checkbox.
4 In the tree, select **Component 1 (comp1)** > **Deformed Geometry, Controls material frame** > **Prescribed Deformation, Shell**.
5 Right-click and choose **Disable**.