Created in COMSOL Multiphysics 6.2

Inverse Uncertainty Quantification of Tensile Test

Introduction

This example demonstrates how to perform inverse uncertainty quantification analysis to calibrate Young's modulus and Poisson's ratio based on a tensile test. The test measures the radial displacement and the tensile force at specified locations of the solid specimen based on different axial displacements. The test data is generated with synthetic data generated in the model. The posterior distributions of the calibrated Young's modulus and Poisson's ratio are computed based on their prior distributions and the test data.

Model Definition

The model run inverse uncertainty quantification study using the radial displacement and the tensile force at specified locations of the solid specimen as the quantities of interests. In order to perform the uncertainty quantification analysis, three random variables need to be defined as the input parameters. First, the prior knowledge of Young's modulus and Poisson's ratio that need to be calibrated. Second, the experimental parameter, the axial displacement, which is the same as the parameter used to generate the test data. The input parameter distributions are set according to the table in Figure 1.

Parameter	Source type	Parameter description
E0 (Young's modulus)	Analytic	Calibration parameter, Normal(3e+11, 5e+10) from [1.4549E11, 4.5451E11]
nu0 (Poisson's ratio)	Analytic	Calibration parameter, Uniform from [0.2, 0.5]
movez (Axial displacement)	Analytic	Experimental parameter, Uniform from [minMove - eps, maxMove + eps]

Figure 1: Input-parameter distributions used in inverse uncertainty quantification.

This tutorial begins with the stationary study for generating the tensile test data. In this analysis, the specimen is assumed to have prescribed displacement in z-direction, where the specified displacement is considered as the experimental parameter. The test data is first generated assuming a Young's modulus value of 200 GPa and a Poisson's ratio of 0.3. The data is then perturbed with noise generated from a Gaussian distribution using a global evaluation. For details, see the Modeling Instructions section.

The experimental data settings in the UQ study uses the tensile test data as the experiential data table. The data column type is specified according to the table in Figure 2.

Experimental Data Settings

Experimental data table: Table 2 (tbl2)

Columns	Type	Settings
movez (m)	Experimental parameter	movez
r_change + errorL*rn1(1+movez) (m)	Quantity of interest	comp1.r_change
tensile_force + errorF*rn1(movez) (N)	Quantity of interest	comp1.tensile_force

Figure 2: Experimental data settings used in inverse uncertainty quantification.

Results and Discussion

The uncertainty quantification study gives the joint probability distribution plot shown in Figure 3, with associated confidence interval information in the calibrated confidence interval table shown in Figure 4. The posterior means of Young's modulus and Poisson's ratio are close to their values used to generate the tensile test data. From the joint probability distribution plot, there is no strong correlation between Young's modulus and Poisson's ratio.

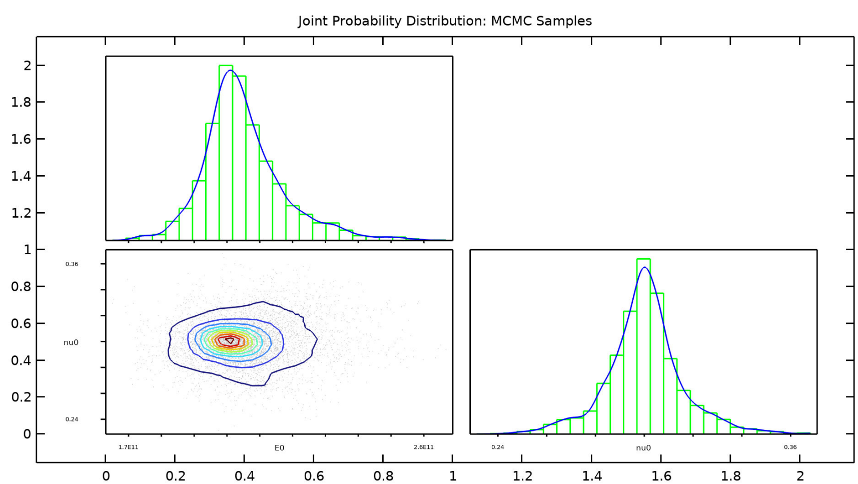
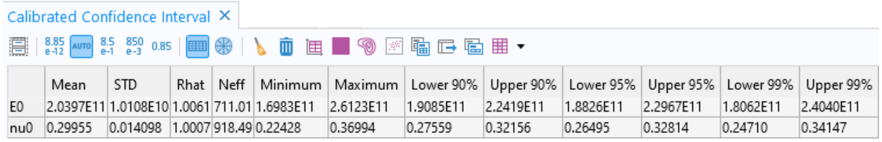


Figure 3: Joint probability distribution.


Calibrated Confidence Interval

	Mean	STD	Rhat	Neff	Minimum	Maximum	Lower 90%	Upper 90%	Lower 95%	Upper 95%	Lower 99%	Upper 99%
E0	2.0397E11	1.0108E10	1.0061	711.01	1.6983E11	2.6123E11	1.9085E11	2.2419E11	1.8826E11	2.2967E11	1.8062E11	2.4040E11
nu0	0.29955	0.014098	1.0007	918.49	0.22428	0.36994	0.27559	0.32156	0.26495	0.32814	0.24710	0.34147

Figure 4: Calibrated confidence interval.

Application Library path: `Uncertainty_Quantification_Module/Tutorials/tensile_test_uncertainty_quantification`

Modeling Instructions

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

NEW

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

GLOBAL DEFINITIONS

Parameters 1

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
movez	1 [mm]	0.001 m	Axial displacement
dispScale	2 [um]	2E-6 m	Radial displacement scale
forceScale	100 [kN]	1E5 N	Tensile force scale
errorF	0.1*forceScale	10000 N	Radial displacement error
errorL	0.5*dispScale	1E-6 m	Tensile force error
E0	200 [GPa]	2E11 Pa	Young's modulus
nu0	0.3	0.3	Poisson's ratio
minMove	200 [um]	2E-4 m	Minimum stretch

Name	Expression	Value	Description
maxMove	2000[um]	0.002 m	Maximum stretch
moveStep	50[um]	5E-5 m	Stretch increment

Geometrical Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Geometrical Parameters` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
r1	5 [mm]	0.005 m	Center radius
r2	10 [mm]	0.01 m	End radii
L1	50 [mm]	0.05 m	Center length
L2	30 [mm]	0.03 m	End length
r3	15 [mm]	0.015 m	Fillet radius
L3	L1/2-sin(pi/4)*r3	0.014393 m	Center length with r1 radius

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **2D Axisymmetric**.

DEFINITIONS

Random 1 (rn1)

1 In the **Home** toolbar, click **Functions** and choose **Local>Random**.

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

3 From the **Distribution** list, choose **Normal**.

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 **mm**.

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `r1`.

4 In the **Height** text field, type `L1/2`.

Rectangle 2 (r2)

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

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 In the **z** text field, type `L1/2`.

4 Locate the **Size and Shape** section. In the **Width** text field, type `r2`.

5 In the **Height** text field, type `L2`.

Circular Arc 1 (ca1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.

2 In the **Settings** window for **Circular Arc**, locate the **Properties** section.

3 From the **Specify** list, choose **Endpoints and radius**.

4 Locate the **Starting Point** section. In the **r** text field, type `r2`.

5 In the **z** text field, type `L1/2`.

6 Locate the **Endpoint** section. In the **r** text field, type `r1`.

7 In the **z** text field, type `L3`.

8 Locate the **Radius** section. In the **Radius** text field, type `r3`.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 In the **Settings** window for **Convert to Solid**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 Locate the **Input** section. Click to clear the **Activate Selection** toggle button for **Input objects**.

5 In the **Model Builder** window, click **Convert to Solid 1 (csol1)**.

6 Click the **Add to Selection** button for **Input objects**.

7 Select the object **r1** only.

8 Click the **Remove from Selection** button for **Input objects**.

9 Select the object **r1** only.

10 In the list, select **r1**.

11 Click the **Remove from Selection** button for **Input objects**.

12 Click in the **Graphics** window and then press Ctrl+A to select all objects.

13 Click to clear the **Activate Selection** toggle button for **Input objects**.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

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

3 From the **Input objects** list, choose **Convert to Solid 1**.

4 Clear the **Keep interior boundaries** check box.

5 Click to clear the **Activate Selection** toggle button for **Input objects**.

COMPONENT 1 (COMP1)

From the **Home** menu, choose **Add Physics**.

ADD PHYSICS

1 Go to the **Add Physics** window.

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

3 Click **Add to Component 1** in the window toolbar.

4 From the **Home** menu, choose **Add Physics**.

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>View 1** node.

2 Right-click **Component 1 (comp1)>Materials** and choose **Blank Material**.

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

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	E0	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	nu0	1	Young's modulus and Poisson's ratio
Density	rho	7850	kg/m³	Basic

SOLID MECHANICS (SOLID)

Symmetry Plane 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **More Constraints>Symmetry Plane**.

Prescribed Displacement 1

In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

Symmetry Plane 1

Select Boundary 2 only.

Prescribed Displacement 1

1. In the **Model Builder** window, click **Prescribed Displacement 1**.
2. Select Boundary 4 only.

DEFINITIONS

Domain Point Probe 1

1. In the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.
2. In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.
3. In row **Coordinates**, set **r** to `r1`.

Point Probe Expression 1 (ppb1)

1. In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.
2. In the **Settings** window for **Point Probe Expression**, locate the **Expression** section.
3. In the **Expression** text field, type `-u`.
4. From the **Table and plot unit** list, choose **m**.
5. In the **Variable name** text field, type `r_change`.

Boundary Probe 1 (bnd1)

1. In the **Definitions** toolbar, click **Probes** and choose **Boundary Probe**.
2. In the **Settings** window for **Boundary Probe**, type `tensile_force` in the **Variable name** text field.
3. Locate the **Probe Type** section. From the **Type** list, choose **Integral**.
4. Locate the **Source Selection** section. From the **Selection** list, choose **Manual**.
5. Select Boundary 4 only.
6. Locate the **Expression** section. In the **Expression** text field, type `solid.sz`.

7 Click to expand the **Table and Window Settings** section.

SOLID MECHANICS (SOLID)

1 Click the **Show More Options** button in the **Model Builder** toolbar.

2 In the **Show More Options** dialog box, select **Physics>Advanced Physics Options** in the tree.

3 In the tree, select the check box for the node **Physics>Advanced Physics Options**.

4 Click **OK**.

5 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

6 In the **Settings** window for **Solid Mechanics**, click to expand the **Advanced Settings** section.

7 Find the **Group ODE variables in solver** subsection. Clear the **Rigid materials** check box.

Prescribed Displacement 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Prescribed Displacement 1**.

2 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

3 From the **Displacement in z direction** list, choose **Prescribed**.

4 In the u_{0z} text field, type `movez`.

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 **General Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

STUDY 1

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Study Settings** section.

2 Select the **Include geometric nonlinearity** check box.

3 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
movez (Axial displacement)	`range(minMove, moveStep, maxMove)`	m

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

RESULTS

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
`r_change + errorL*rn1(1+movez)`	m	
`tensile_force + errorF*rn1(movez)`	N	

4 Click **Evaluate**.

ADD STUDY

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

2 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

3 Click **Add Study** in the window toolbar.

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

STUDY 2

Uncertainty Quantification

In the **Study** toolbar, click **Uncertainty Quantification**.

Step 1: Stationary

1 In the **Model Builder** window, click **Step 1: Stationary**.

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

3 Select the **Include geometric nonlinearity** check box.

Uncertainty Quantification

1 In the **Model Builder** window, click **Uncertainty Quantification**.

2 In the **Settings** window for **Uncertainty Quantification**, locate the **Uncertainty Quantification Settings** section.

3 From the **UQ study type** list, choose **Inverse uncertainty quantification**.

4 Locate the **Quantities of Interest** section. Click + **Add**.

5 In the table, enter the following settings:

Expression	Description	Individual solution to use
`comp1.r_change`		From "Solution to use"

6 Click + **Add**.

7 In the table, enter the following settings:

Expression	Description	Individual solution to use
`comp1.tensile_force`		From "Solution to use"

8 Locate the **Uncertainty Quantification Settings** section. Find the **Surrogate model settings** subsection. From the **Surrogate model** list, choose **Adaptive sparse polynomial chaos expansion**.

9 Locate the **Input Parameters** section. Click + **Add**.

10 In the table, enter the following settings:

Parameter	Source type	Parameter description
E0 (Young's modulus)	Analytic	Calibration parameter, Normal

11 From the **Distribution** list, choose **Normal(μ,σ)**.

12 In the **Mean** text field, type `3e+11`.

13 In the **Standard deviation** text field, type `5e+10`.

14 Click + **Add**.

15 In the table, enter the following settings:

Parameter	Source type	Parameter description
nu0 (Poisson's ratio)	Analytic	Calibration parameter, Uniform

16 In the **Lower bound** text field, type `0.2`.

17 In the **Upper bound** text field, type `0.5`.

18 Click + **Add**.

19 In the table, enter the following settings:

Parameter	Source type	Parameter description
movez (Axial displacement)	Analytic	Calibration parameter, Uniform

20 In the **Lower bound** text field, type `minMove - eps`.

21 In the **Upper bound** text field, type `maxMove + eps`.

22 Locate the **Experimental Data Settings** section. From the **Experimental data table** list, choose **Table 2**.

23 In the table, enter the following settings:

Columns	Type	Settings
movez (m)	Experimental parameter	movez

24 From the **Name** list, choose **movez**.

25 In the table, click to select the cell at row number 2 and column number 3.

26 From the **Name** list, choose **comp1.r_change**.

27 In the table, click to select the cell at row number 3 and column number 3.

28 From the **Name** list, choose **comp1.tensile_force**.

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

RESULTS

Stress, 3D (solid) 1

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