AIMS Energy, 12(3): 561–599.
DOI: 10.3934/energy.2024027
Received: 25 February 2024
Revised: 08 April 2024
Accepted: 28 April 2024
Published: 15 May 2024

https://www.aimspress.com/journal/energy

Research article

Design analysis and simulation of serpentine-shaped piezoelectric cantilever beam for pipeline vibration-based energy harvester

Wan Nabila Mohd Fairuz*, Illani Mohd Nawi, Mohamad Radzi Ahmad and Ramani Kannan

Department of Electrical and Electronic Engineering, Universiti Teknologi PETRONAS, 32610 Seri Iskandar, Perak, Malaysia

* **Correspondence:** Email: wan_20002005@utp.edu.my; Tel: +60175925810.

Supplementary

COMSOL MULTIPHYSICS

MODELLING 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** > **Electromagnetics-Structural Mechanics Interaction** > **Piezoelectricity** > **Piezoelectricity, Solid**.

3. Click **Add**.
4. In the **Select Physics** tree, select **AC/DC > Electrical Circuit (cir)**.
5. Click **Add**.
6. In the **Select Study** tree, select **General Studies > Frequency Domain**.
7. Click **Done**.

Note: The **Model Builder** will appear on the screen. The **Physics** and **Study** can be added manually through the **Home** toolbar.

8. From the **Home** toolbar, click **Add Study**.
9. In the **Add Study** tree, select **Preset Studies for Selected Multiphysics > Eigenfrequency**.
10. Click **Add Study**.

GLOBAL DEFINITIONS

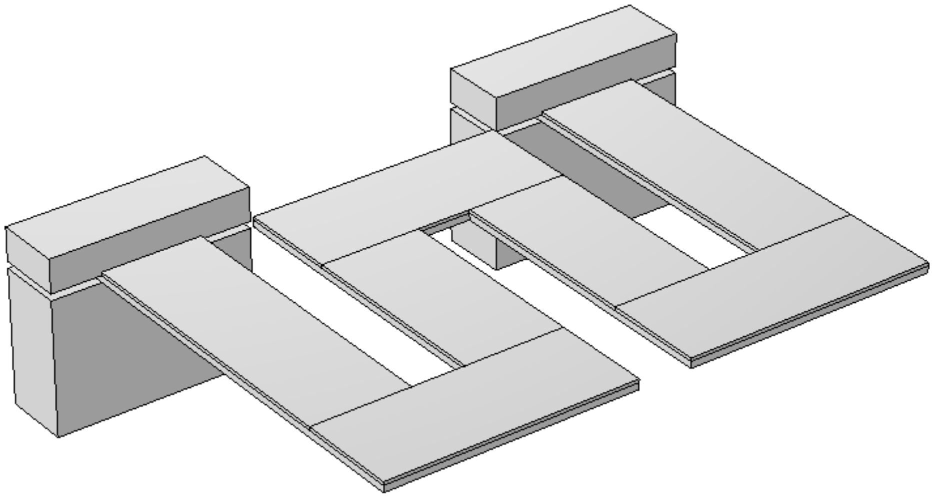
Parameters 1

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 parameters:

Name	Expression	Value	Description
width	1.5 [cm]	0.015 m	piezo width (x axis)
p14length	4.5 [cm]	0.045 m	piezo length (y axis)—beam 1 & 4
p23length	3 [cm]	0.03 m	piezo length (y axis) —beam 2 & 3
thickness	0.05 [cm]	5E−4 m	piezo thickness (z axis)
gap	0.8 [cm]	0.008 m	gap between next beam (x axis)
connector_length	width	0.015 m	connector between beam length (x axis)
connector_width	Width + gap + width	0.038 m	connector between beam width (y axis)
top_anchor_thickness	0.5 [cm]	0.005 m	top anchor height (z axis)
anchor_width	1 [cm]	0.01 m	anchor width (y axis)
anchor_length	Width + 1.5 [cm]	0.03 m	anchor length (x axis)
lower_anchor_thickness	2 [cm]	0.02 m	lower anchor height (z axis)
clamp	1 [cm]	0.01 m	clearance to clamp beam (y axis)
beam	0.1 [cm]	0.001 m	non piezo thickness (z axis)
acc	1	1	acceleration (g)
r_load	31 [kΩ]	31000 Ω	load resistance

Note: The parameters can be edited through **Global Definitions > Parameters 1** instead of changing it manually on each section.

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 **cm**.

Block 1 (blk1)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as 'piezo 1', and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type 'width'.
5. In the **Depth** text field, type 'p14length'.
6. In the **Height** text field, type 'thickness'.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type '0'.
9. In the **y** text field, type 'connector_length'
10. In the **z** text field, type 'beam'

11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 2 (blk2)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘piezo 2’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type ‘width’.
5. In the **Depth** text field, type ‘p23length’.
6. In the **Height** text field, type ‘thickness’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘width + gap’.
9. In the **y** text field, type ‘connector_length’
10. In the **z** text field, type ‘beam’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 3 (blk3)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘piezo 3’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type ‘width’.
5. In the **Depth** text field, type ‘p23length’.
6. In the **Height** text field, type ‘thickness’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘(width*2) + (gap*2)’.

9. In the **y** text field, type ‘connector_length’
10. In the **z** text field, type ‘beam’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 4 (blk4)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘piezo 4’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type “width’.
5. In the **Depth** text field, type ‘p14length’.
6. In the **Height** text field, type ‘thickness’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘(width*3) + (gap*3)’.
9. In the **y** text field, type ‘connector_length’
10. In the **z** text field, type ‘beam’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 5 (blk5)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘piezo connector 1’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type “connector_width’.
5. In the **Depth** text field, type ‘connector_length’.
6. In the **Height** text field, type ‘thickness’.

7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘0’.
9. In the **y** text field, type ‘0’
10. In the **z** text field, type ‘beam’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 6 (blk6)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘piezo connector 2’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type “connector_width’.
5. In the **Depth** text field, type ‘connector_length’.
6. In the **Height** text field, type ‘thickness’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘width + gap’.
9. In the **y** text field, type ‘p23length + connector_length’
10. In the **z** text field, type ‘beam’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 7 (blk7)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘piezo connector 3’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type “connector_width’.

5. In the **Depth** text field, type ‘connector_length’.
6. In the **Height** text field, type ‘thickness’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘(width*2) + (gap*2)’.
9. In the **y** text field, type ‘0’
10. In the **z** text field, type ‘beam’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Union 1 (uni1)

1. In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union.**
2. In the **Settings** for **Union**, locate **Input objects** and enable the button to green.
3. Click the Zoom to Selection button in the Graphic.
4. Select **blk1, blk2, blk3, blk4, blk5, blk6** and **blk7** in the **Graphic** and make sure it appears in the **Input objects** selection.
5. Clear the **Keep input objects** and **keep interior boundaries** check box.
6. Click **Build All Objects**.

Block 8 (blk8)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘beam 1’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type ‘width’.
5. In the **Depth** text field, type ‘p14length + clamp’.
6. In the **Height** text field, type ‘beam’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘0’.

9. In the **y** text field, type ‘connector_length’
10. In the **z** text field, type ‘0’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 9 (blk9)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘beam 2’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type ‘width’.
5. In the **Depth** text field, type ‘p23length’.
6. In the **Height** text field, type ‘beam’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘width + gap’.
9. In the **y** text field, type ‘connector_length’
10. In the **z** text field, type ‘0’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 10 (blk10)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘beam 3’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type ‘width’.
5. In the **Depth** text field, type ‘p23length’.
6. In the **Height** text field, type ‘beam’.

7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘(width*2) + (gap*2)’.
9. In the **y** text field, type ‘connector_length’
10. In the **z** text field, type ‘0’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 11 (blk11)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘beam 4’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type “width’.
5. In the **Depth** text field, type ‘p14length + clamp’.
6. In the **Height** text field, type ‘beam’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘(width*3) + (gap*3)’.
9. In the **y** text field, type ‘connector_length’
10. In the **z** text field, type ‘0’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 12 (blk12)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘beam connector 1’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type “connector_width’.

5. In the **Depth** text field, type ‘connector_length’.
6. In the **Height** text field, type ‘beam’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘0’.
9. In the **y** text field, type ‘0’
10. In the **z** text field, type ‘0’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 13 (blk13)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘beam connector 2’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type “connector_width’.
5. In the **Depth** text field, type ‘connector_length’.
6. In the **Height** text field, type ‘beam’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘width + gap’.
9. In the **y** text field, type ‘p23length + connector_length’
10. In the **z** text field, type ‘0’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 14 (blk14)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘beam connector 3’, and ensure that the **Object Type** is **Solid**.

3. Locate the **Size and Shape** section.
4. In the **Width** text field, type “connector_width’.
5. In the **Depth** text field, type ‘connector_length’.
6. In the **Height** text field, type ‘beam’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘(width*2) + (gap*2)’.
9. In the **y** text field, type ‘0’
10. In the **z** text field, type ‘0’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Union 2 (uni2)

1. In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union.**
2. In the **Settings** for **Union**, locate **Input objects** and enable the button to green.
3. Click the Zoom to Selection button in the Graphic.
4. Select **blk8, blk9, blk10, blk11, blk12, blk13** and **blk14** in the **Graphic** and make sure it appears in the **Input objects** selection.
5. Clear the **Keep input objects** and **keep interior boundaries** check box.
6. Click **Build All Objects**.

Block 15 (blk15)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘anchor down 1’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type “constraint_width’.
5. In the **Depth** text field, type ‘constraint_length’.
6. In the **Height** text field, type ‘lower_constraint_thickness’.

7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type '−0.75'.
9. In the **y** text field, type 'connector_length + p14length'
10. In the **z** text field, type '-lower_constraint_thickness'
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 16 (blk16)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as 'anchor down 2', and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type "constraint_width'.
5. In the **Depth** text field, type 'constraint_length'.
6. In the **Height** text field, type 'lower_constraint_thickness'.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type '(width*3) + (gap*3) −0.75'.
9. In the **y** text field, type 'connector_length + p14length'
10. In the **z** text field, type '-lower_constraint_thickness'
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Block 17 (blk17)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as 'anchor up 1', and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type "constraint_width'.

5. In the **Depth** text field, type ‘constraint_length’.
6. In the **Height** text field, type ‘constraint_thickness’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘−0.75’.
9. In the **y** text field, type ‘connector_length + p14length’
10. In the **z** text field, type ‘beam’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

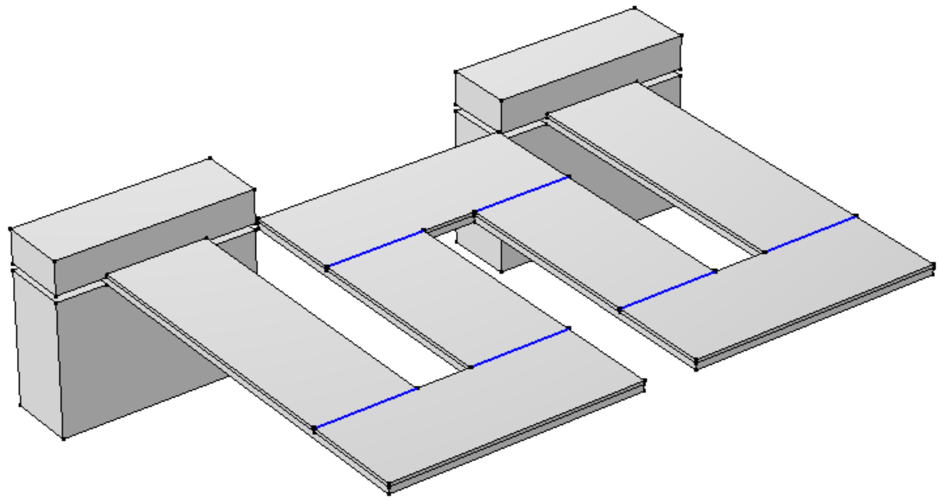
Block 18 (blk18)

1. In the **Geometry** toolbar, click **Block**.
2. In the **Settings** window for Block, **label** the block as ‘anchor up 2’, and ensure that the **Object Type** is **Solid**.
3. Locate the **Size and Shape** section.
4. In the **Width** text field, type “constraint_width’.
5. In the **Depth** text field, type ‘constraint_length’.
6. In the **Height** text field, type ‘constraint_thickness’.
7. Locate the **Position** section, and ensure that the **Base** is **Corner**.
8. In the **x** text field, type ‘(width*3) + (gap*3) − 0.75’.
9. In the **y** text field, type ‘connector_length + p14length’
10. In the **z** text field, type ‘beam’
11. Locate the **Axis** section, and change the **Axis type** to **z-axis**.
12. Click **Build All Objects**.

Ignore Edges 1 (ige1)

1. In the **Geometry** toolbar, select **Virtual Operations** and click **Ignore Edges**.
2. In the **Settings** for **Ignore Edges**, locate **Edges to ignore** and enable the button to green.
3. Click the **Zoom to Selection** button in the **Graphic**.

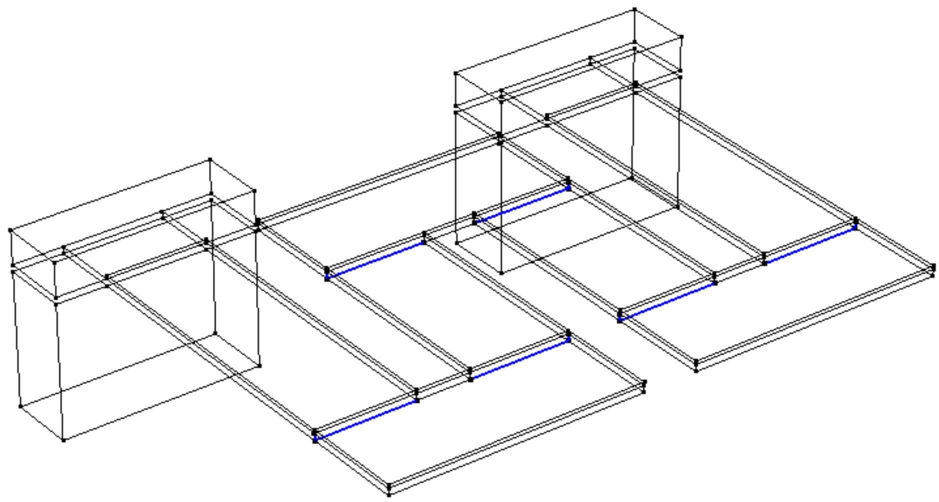
4. On the object uni1, select all unwanted all unwanted edges for piezoelectric layer.



5. Click on the **Ignore adjacent vertices** check box.
6. Ensure that the green button is enabled and click **Build All**.

Ignore Edges 2 (ige2)

1. In the **Geometry** toolbar, select **Virtual Operations** and click **Ignore Edges**.
2. In the **Settings** for **Ignore Edges**, locate **Edges to ignore** and enable the button to green.
3. Click the **Wireframe Rendering** button in the **Graphic**.
4. Click the **Zoom to Selection** button in the **Graphic**.
5. On the object uni2, select all unwanted all unwanted edges for beam layer.



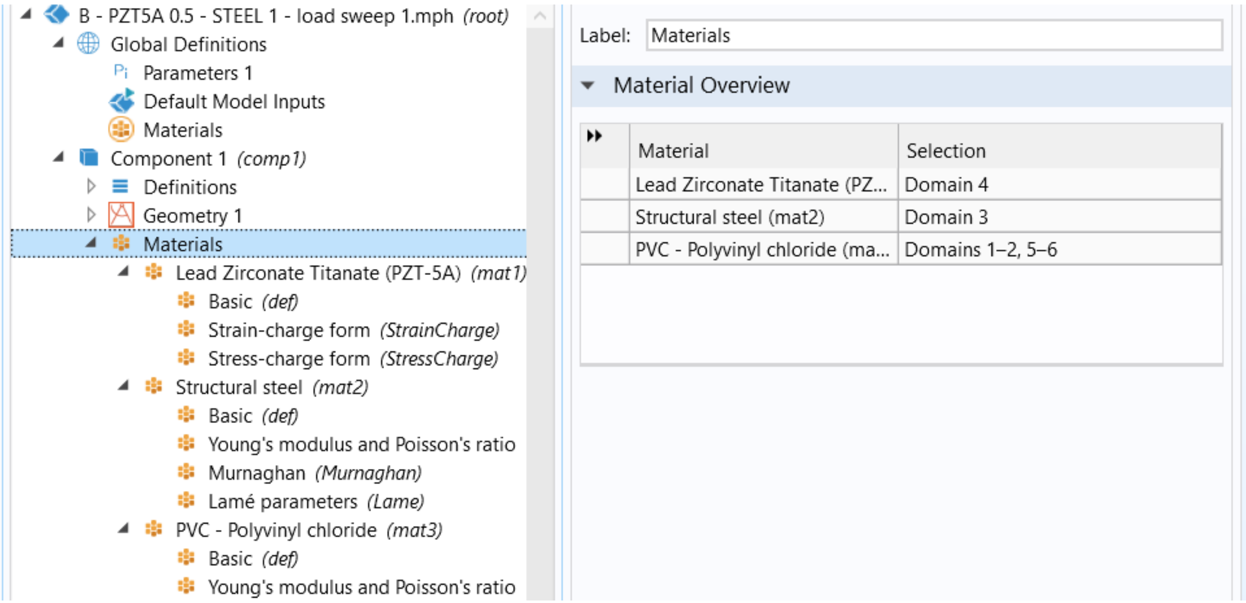
6. Click on the **Ignore adjacent vertices** check box.
7. Ensure that the green button is enabled and click **Build All**.

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, search for **Lead Zirconate Titanate (PZT)**.
4. Select **Lead Zirconate Titanate (PZT-5A)** as **mat1**.
5. Click **Add to Component** in the window toolbar.
6. In the tree, search for **Steel**.
7. Select **Structural Steel** as **mat2**.
8. Click **Add to Component** in the window toolbar.
9. In the tree, search for **PVC**.
10. Select **Polyvinyl Chloride (PVC)** as **mat3**.
11. Click **Add to Component** in the window toolbar.
12. Close the **Add Material** window.
13. In the **Model Builder** under **Component 1** tree, select **Materials**.
14. In the **Settings** for **Lead Zirconate Titanate (PZT-5A),** identify **Geometry Entity Selection.**
15. For **Geometry entity level**, select **Domain**.
16. For **Selection**, select **Manual**.
17. Select **uni1 (Domain 4)** in the **Graphic** as piezoelectric layer.
18. In the **Settings** for **Structural Steel,** identify **Geometry Entity Selection.**
19. For **Geometry entity level**, select **Domain**.
20. For **Selection**, select **Manual**.
21. Select **uni2 (Domain 3)** in the **Graphic** as non-piezoelectric beam layer.
22. In the **Settings** for **Polyvinyl Chloride (PVC),** identify **Geometry Entity Selection.**
23. For **Geometry entity level**, select **Domain**.
24. For **Selection**, select **Manual**.

25. Select **blk15, blk16, blk17** and **blk18 (Domain 1, 2, 5** and **6)** in the **Graphic**.

26. Ensure that all the blocks are assigned with materials in the **Material Overview**.

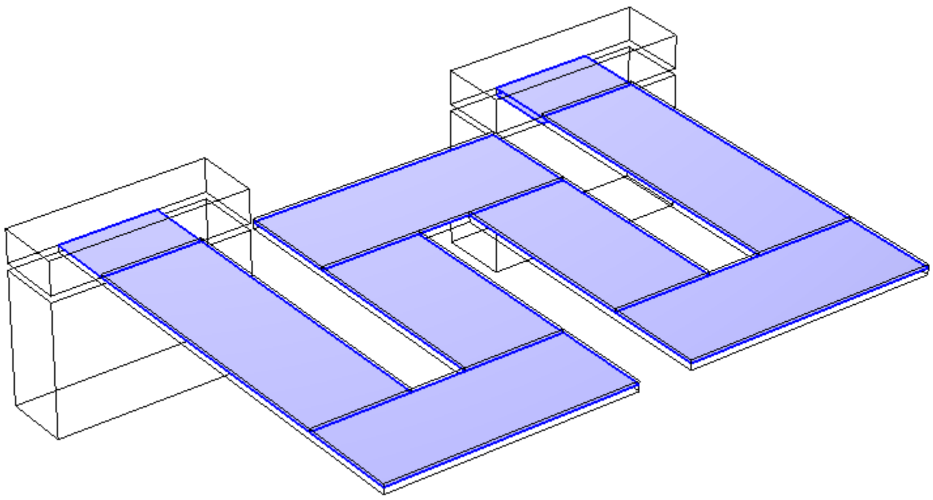


<u>ELECTROSTATICS (*es*)</u>

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Electrostatics (es)**.
2. Select **Domain 3** and **4** only.

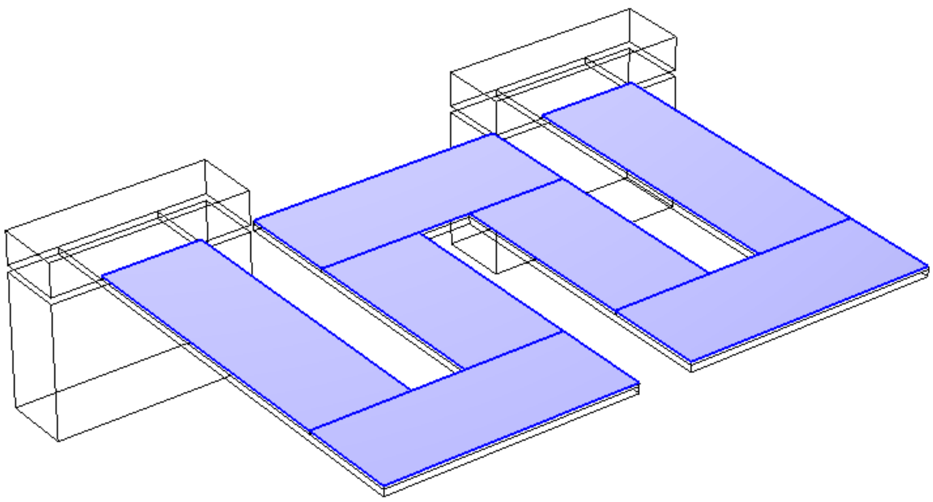
Ground

1. In the **Physics** toolbar, click **Boundaries** and choose **Ground**.
2. In the **Ground 1 Setting**, identify the **Boundary Selection** and set it to **Manual Selection**.
3. Enable the button until it turned green.
4. Click on **Zoom to Selection** in the **Graphic**.
5. Select all the lower surface of piezoelectric layer as the **Ground** boundaries.



Terminal

1. In the **Physics** toolbar, click **Boundaries** and choose **Terminal**.
2. In the **Terminal 1 Setting**, identify the **Boundary Selection** and set it to **Manual Selection**.
3. Enable the button until it turned green.
4. Click on **Zoom to Selection** in the **Graphic**.
5. Select all the upper surface of piezoelectric layer as the **Terminal** boundaries.



6. In the **Settings** window for **Terminal 1**, locate the **Terminal name** section and type '1'.
7. From the **Terminal type** list, choose **Circuit**.

ELECTRICAL CIRCUIT (*cir*)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electrical Circuit (cir)**.

Ground Node 1 (gnd1)

1. Click **Ground Node 1 (gnd1)** in the **Electrical Circuit (cir)** selection tree.
2. In the **Setting** for the **Ground Node 1**, locate **Node Connections** section.
3. In the table, enter the following settings:

Label	Node names
p	0

Resistor 1 (R1)

1. In the **Electrical Circuit** toolbar, click **Resistor**.
2. In the **Settings** window for **Resistor**, locate the **Node Connections** section.
3. In the table, enter the following settings:

Label	Node names
p	1
n	0

4. Locate the **Device Parameters** section. In the **R** text field, type 'R_load'.

Note: Resistor is only for short-circuit test. Replace resistor with voltmeter for open-circuit test.

Voltmeter 1 (vm1)

1. In the **Electrical Circuit** toolbar, click **Volt Meter**.
2. In the **Settings** window for **Volt Meter 1 (vm1)**, locate the **Node Connections** section.
3. In the table, enter the following settings:

Label	Node names
p	1
n	0

Note: Voltmeter is only for open-circuit test. Replace voltmeter with resistor for short-circuit test.

External I-Terminal 1 (termI 1)

1. In the **Electrical Circuit** toolbar, click **External I-Terminal**.
2. In the **Settings** window for **External I-Terminal**, locate the **Node Connections** section.
3. In the **Node name** text field, type '1'.
4. Locate the **External Terminal** section. From the **V** list, choose **Terminal voltage (es/term1)**.

SOLID MECHANICS (*solid*)

Linear Elastic Material 1

In the Model Builder window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Damping 1

1. In the **Physics** toolbar, click **Attributes** and choose **Damping**.
2. In the **Settings** window for **Damping**, locate the **Damping Settings** section.
3. From the **Damping type** list, choose **Isotropic loss factor**.
4. From the $\boldsymbol{\eta_s}$ list, choose **User defined**. In the associated text field, type '0.001'.

Free 1

The **Setting** for **Free 1** is selected as default by the COMSOL software.

Initial Values 1

The **Setting** for **Initial Values 1** is selected as default by the COMSOL software.

Piezoelectric Material 1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Piezoelectric Material 1**.
2. In the **Settings** window for **Piezoelectric Material**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Manual**.

4. Select **Domain 4**.

Mechanical Damping 1

1. In the **Physics** toolbar, click **Attributes** and choose **Mechanical Damping**.
2. 2 In the **Settings** window for **Mechanical Damping**, locate the **Damping Settings** section.
3. From the **Damping type** list, choose **Isotropic loss factor**.
4. From the $\boldsymbol{\eta_s}$ list, choose **User defined**. In the associated text field, type '0.001'.

Fixed Constraint 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
2. In the **Settings** window for **Fixed Constraint**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Manual**.
4. Select **Domain 1,2,5**, and **6**.

Body Load 1

1. In the **Physics** toolbar, click **Domains** and choose **Body Load**.
2. In the **Settings** window for **Body Load**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **All domains**.
4. Locate the **Force** section. Select **Load type** as **Force per unit volume**.
5. Change $\boldsymbol{F_v}$ as **User defined** and specify the $\boldsymbol{F_v}$ vector as

0	x
0	y
-Solid.rho * g_const * acc	z

MESH 1

1. In the **Model Builder** window, click **Mesh**.
2. In the **Settings** window for **Study**, locate **Sequence Type** and select **Physics-controlled mesh**.
3. In the **Physics-Controlled Mesh** setting, locate **Element size** and select **Finer**.

4. Click **Build All** to create mesh on the geometry.

Note: Smaller element size will increase the accuracy but will increase the running time.

FREQUENCY RESPONSE—Study 1

1. In the **Model Builder** window, click **Study 1**.
2. In the **Settings** window for **Study**, type 'Frequency Response: Voltage and Power' in the **Label** text field.

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Frequency Response** click **Step 1: Frequency Domain**.
2. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
3. In the **Frequencies** text field, type 'range (10,1,300)', where 10 is the minimum frequency, 1 is the interval or step size and 300 is the maximum frequency.
4. Click **Compute** to run the study.

Note: Change the interval to lower size to get more accurate value but it will increase running time. Change the minimum and maximum frequency value to closer range to get better view and closer value.

RESULTS—Study 1

1. In the **Model Builder** window, under **Result**, the **Dataset 1** based on the study will be generated.
2. From the dataset, results such as El**ectric Potential (es), Stress (solid), Applied Loads (solid)** and **Electric Field Norm (es)** will be generated by default.

1D Plot—Frequency Response: Voltage and Power

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type 'Frequency Response: Voltage & Power' in the **Label** text field.

3. Click to expand the **Title** section. From the **Title** type list, choose **Manual**.
4. In the **Title** text area, type ‘Frequency Response: Voltage & power’.

Global 1

1. Right-click **Frequency Response: Voltage & Power** and choose **Global**.
2. In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3. In the table, enter the following settings:

Expression	Unit	Description
Abs (cir. R1_v)	V	Voltage (V)
0.5*Realdot (cir. R1_i, cir. R1_v)	mW	Electric power out (mW)

4. Locate **Legends** in the settings, click check on **Shows Legends** check box.
5. For **Legends**, select **Automatic**.

Graph Marker 1

1. Right-click **Global 1** and choose **Graph Marker**.
2. In the **Settings** window for **Graph Marker 1**, locate the **Display** section.
3. Select **Display mode** as **Min and max**. Choose the **Display** as **Max** and **Scope** as **Global**.
4. Locate **Text Format** in the settings, click check on **Show x-coordinate** check box.
5. In the **Frequency Response: Voltage & Power** toolbar, click **Plot**.

ADD STUDY

1. In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add S**tudy window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Frequency Domain**.
4. Click **Add Study** in the window toolbar and close the **Add Study** window.
5. The new study will appear on the **Model Builder**.

LOAD DEPENDENCY—Study 2

1. In the **Model Builder** window, click **Study 2**.
2. In the **Settings** window for **Study 2**, type ‘Load Dependency’ in the **Label** text field.

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Load Dependency** click **Step 1: Frequency Domain**.
2. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
3. In the **Frequencies** text field, type ’262.65’, where 262.65 is the resonant frequency obtained from **Study 1**.
4. Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.
5. Locate **Sweep type** and change to **All combinations**.
6. Click **+** to add new parameter.
7. In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
R_load	Range (250,250,1000000)	Ω

8. In the Model Builder window, click **Load Dependence**.
9. In the **Settings** window for **Load Dependence**, locate the **Study Settings** section.
10. Clear the **Generate default plots** check box.
11. Click **Compute** to run the study.

RESULTS—Study 2

In the **Model Builder** window, under **Result**, the **Dataset 2** will be generated.

1D Plot—Load Dependency: Voltage and Power

1. In the **Model Builder** window, under **Results** right-click **Frequency Response: Voltage & Power** and choose **Duplicate**.

2. In the **Settings** window for **Frequency Response: Voltage & Power 1**, type ‘Load Dependency: Voltage & Power’ in the **Label** text field.
3. Locate the **Data** section. From the **Dataset list**, choose **Load Dependence/Solution 2 (sol2)**.
4. Click to expand the **Title** section. From the **Title** type list, choose **Manual**. In the **Title** text area, type ‘Load Dependency: Voltage & Power’.
5. Locate the **Legend** section. From the **Position list**, choose **Upper left**.
6. In the **Load Dependence: Voltage & Power** toolbar, click **Plot**.

ADD STUDY

1. In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add S**tudy window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Frequency Domain**.
4. Click **Add Study** in the window toolbar and close the **Add Study** window.
5. The new study will appear on the **Model Builder**.

ACCELERATION DEPENDENCY—Study 3

1. In the **Model Builder** window, click **Study 3**.
2. In the **Settings** window for **Study 3**, type ‘Acceleration Dependency’ in the **Label** text field.

Step 1: Frequency Domain

1. In the **Model Builder** window, under **Acceleration Dependency** click **Step 1: Frequency Domain**.
2. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
3. In the **Frequencies** text field, type ’262.65’, where 262.65 is the resonant frequency obtained from **Study 1**.
4. Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.
5. Locate **Sweep type** and change to **All combinations**.

6. Click + to add new parameter.
7. In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Acc	0.5, 1, 1.5, 2	1

8. In the Model Builder window, click **Acceleration Dependence**.
9. In the **Settings** window for **Acceleration Dependence**, locate the **Study Settings** section.
10. Clear the **Generate default plots** check box.
11. Click **Compute** to run the study.

RESULTS—Study 3

In the **Model Builder** window, under **Result**, the **Dataset 3** will be generated.

1D Plot—Acceleration Dependency: Voltage

1. In the **Model Builder** window, under **Results** right-click **Load Dependency: Voltage & Power** and choose **Duplicate**.
2. In the **Settings** window for **Load Dependency: Voltage & Power 1**, type 'Acceleration Dependency: Voltage & Power' in the **Label** text field.
3. Locate the **Data** section. From the **Dataset list**, choose **Acceleration Dependence/Solution 3 (sol3)**.
4. Click to expand the **Title** section. From the **Title** type list, choose **Manual**. In the **Title** text area, type 'Acceleration Dependency: Voltage'.
5. Locate the **Plot Settings** section.
6. Select the **x-axis label** check box. In the associated text field, type 'Acceleration (g)'.
7. Locate the **Axis** section. Clear the **x-axis log scale** check box.
8. Locate the **Legend** section. From the **Position list**, choose **Upper left**.

Global 1

1. Right-click **Acceleration Dependency: Voltage** and choose **Global**.

2. In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3. In the table, enter the following settings:

Expression	Unit	Description
Abs (cir. R1_v)	V	Voltage (V)

4. Locate the **x-Axis Data**. Change the **Axis source data** to **acc**.
5. Locate **Legends** in the settings, click check on **Shows Legends** check box.
6. For **Legends**, select **Automatic**.
7. In the **Acceleration Dependence: Voltage** toolbar, click **Plot**.

3D Plot—Stress Distribution

1. In the **Model Builder** window, under **Results**, click on **Stress (solid)**.
2. In the **Settings** window for **Stress (solid)**, type 'Stress Distribution' in the **Label** text field.
3. Locate the **Data** section. From the **Dataset list**, choose **Acceleration Dependence/Solution 3 (sol3)**.
4. From the **Parameter value (acc)**, choose **1**. (Graphic figure for other parameter can be seen after changing the value)
5. In the **Graphic**, click on **Go to default view**.
6. Locate the **Selection** section. For the **Geometric entity level,** choose **Domain**.
7. Choose **Manual** Selection and select **Domain 3** and **4** in the **Graphic**.
8. Click to expand the **Title** section. From the **Title** type list, choose **Automatic**.
9. Locate the **Color Legend** section. Click check on **Show legends**, **Show maximum and minimum values** and **Show units** check box.
10. On the **Settings** for **Stress Distribution**, click **Plot**.

3D Plot—Electric Potential

1. In the **Model Builder** window, under **Results**, click on **Electric Potential (es)**.
2. In the **Settings** window for **Electric Potential (es)**, type 'Electric Potential' in the **Label** text field.

3. Locate the **Data** section. From the **Dataset list**, choose **Acceleration Dependence/Solution 3 (sol3)**.
4. From the **Parameter value (acc)**, choose **1**. (Graphic figure for other parameter can be seen after changing the value)
5. In the **Graphic**, click on **Go to default view**.
6. Locate the **Selection** section. For the **Geometric entity level,** choose **Domain**.
7. Choose **Manual** Selection and select **Domain 3** and **4** in the **Graphic**.
8. Click to expand the **Title** section. From the **Title** type list, choose **Automatic**.
9. Locate the **Color Legend** section. Click check on **Show legends**, **Show maximum and minimum values** and **Show units** check box.
10. On the **Settings** for **Electric Potential**, click **Plot**.

3D Plot—Displacement Magnitude

1. In the **Model Builder** window, under **Results**, right click on **Stress (solid)**, select **Duplicate**.
2. Click on **Stress (solid) 1**.
3. In the **Settings** window for **Stress (solid) 1**, type 'Displacement Magnitude' in the **Label** text field.
4. Locate the **Data** section. From the **Dataset list**, choose **Acceleration Dependence/Solution 3 (sol3)**.
5. From the **Parameter value (acc)**, choose **1**. (Graphic figure for other parameter can be seen after changing the value)
6. In the **Graphic**, click on **Go to default view**.
7. Locate the **Selection** section. For the **Geometric entity level,** choose **Domain**.
8. Choose **Manual** Selection and select **Domain 3** and **4** in the **Graphic**.
9. Click to expand the **Title** section. From the **Title** type list, choose **Automatic**.
10. Locate the **Color Legend** section. Click check on **Show legends**, **Show maximum and minimum values** and **Show units** check box.
11. In the **Model Builder**, under **Displacement Magnitude**, click on **Surface 1**.

12. In the **Settings** for **Surface**, locate the **Expression** section.
13. Find and click on **Replace Expression** above the **Expression** text field.
14. In the **Replace Expression** selection tree, select **Model > Component 1 (comp1) > Solid Mechanics > Displacement > Solid.disp—Displacement Magnitude—m**.
15. Locate **Unit** and type ‘cm’ in the text field.
16. Locate **Coloring and Style** section, select **Traffic Light Classic** for the **Color table**.
17. On the **Settings** for **Stress Distribution**, click **Plot**.

ADD STUDY

1. In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add S**tudy window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Multiphysics > Eigenfrequency**.
4. Click **Add Study** in the window toolbar and close the **Add Study** window.

EIGENFREQUENCY—Study 4

Eigenfrequency study is conducted using open-circuit, therefore a voltmeter is required instead of resistor.

1. In the **Model Builder** window, click **Study 4**.
2. In the **Settings** window for **Study 4**, type ‘Eigenfrequency’ in the **Label** text field.

Step 1: Eigenfrequency

1. In the **Model Builder** window, under **Eigenfrequency** click **Step 1: Eigenfrequency**.
2. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
3. In the **Desired number of eigenfrequencies** text field, click check and type ‘6’.
4. In the **Unit** text field, type ‘Hz’.

5. In the **Search for eigenfrequencies around** text field, click check and type ’300’, where 300 is the maximum frequency.
6. Click **Compute**.

<u>RESULTS—Study 4</u>

1. In the **Model Builder** window, under **Result**, the **Dataset 4** based on the study will be generated.
2. From the dataset, results such as **Mode Shape (solid)**, **Eigenfrequencies** and **Participation Factors** will be generated by default.
3. In the **Model Builder** window, under **Result**, click on **Eigenfrequencies** to obtain the result.