Model created in COMSOL Multiphysics 6.4

Electrostatic Chuck

Introduction

Electrostatic chucks (e-chucks) play an important role in various wafer-processing equipment. Instead of mechanical clamping, an e-chuck uses an electromechanical force to secure a wafer on a temperature-controlled platform that cools or heats the wafer during processing. Usually, wafers are warped so e-chucks are also used in lithographic tools to flatten wafers during exposure. This model demonstrates the basics of e-chuck operation resulting from the couplings of electrostatic force, gas flow, heat transfer, and solid mechanics in the context of wafer cooling. In this model, an electrostatic force counters the pressure from helium gas flowing in the gap between the wafer and the e-chuck to provide efficient thermal conduction in an otherwise low pressure environment.

Model Definition

This 2D axisymmetric model comprises three regions to represent the wafer (deformable), two dielectric blocks (rigid), and a gas channel. The dielectric blocks are the only parts of the e-chuck that are explicitly included in the geometry. The silicon wafer (diameter = 100 mm) sits on a stand-off at the edge of the outer dielectric block and is suspended above the dielectric blocks, forming a 40 μm-gap. This gap region is specified as a **Fluid Properties** domain, in which the flow of helium gas is solved for. At the edge, the wafer and the stand-off on the e-chuck are in contact but the wafer is able to slide or detach depending on the resultant force on it. The wafer's center is suspended above the e-chuck surface but can come into contact with the e-chuck if the applied bias exceeds the pull-in voltage. To allow for large displacements, a **Deforming Domain** is applied to the gap between the wafer and dielectric blocks. An electromechanical force between the wafer and the e-chuck results from applying **Terminal** boundaries to the bottom of the wafer and the dielectric blocks. The model geometry is shown in Figure 1. The y-axis is scaled by a factor of 20 to better show the narrow gap between the wafer and the e-chuck.

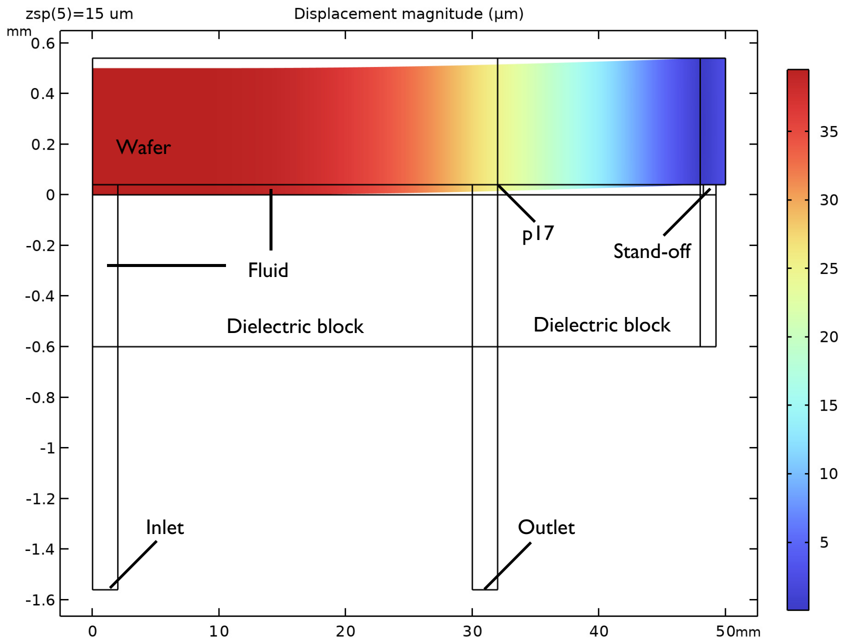


Figure 1: The model geometry.

This model solves for the combinations of applied voltage and gas flow required to keep the wafer securely in place as characterized by the z-displacement of a point on the bottom surface of the wafer positioned 32 mm from the center. This point is labeled `p17` in the geometry. As long as `p17` is below z = 40 µm, the wafer experiences a net downward force. COMSOL Multiphysics can then solve for the DC voltage that must be applied to the wafer to move `p17` to the specified z-coordinate setpoint, or `zsp`. This is achieved by adding a global equation for the DC voltage, `VdcSP`, which is applied at the bottom of the dielectric blocks. The equation `intop1(z)-zsp=0` is then solved to determine the value of `VdcSP`. This means that `VdcSP` is adjusted until `p17` is at `zsp`. Solving the problem in this manner avoids complications when the problem has no solution. The result of the analysis is a plot of displacement versus applied voltage.

Physics Interfaces

The model uses four interfaces: **Laminar Flow**, **Solid Mechanics**, **Heat Transfer in Solids and Fluids** (which are built into the **Fluid-Solid Interaction** from **Conjugate Heating**), and

Electrostatics. With these four interfaces, the available Multiphysics couplings are **Fluid-Structure Interaction**, **Nonisothermal Flow**, and **Electromechanical Forces**.

MATERIAL PROPERTIES

Silicon and aluminum oxide material models are used for the wafer and the dielectrics, respectively. To simulate heat transfer between the wafer and the e-chuck via helium gas, the model applies a user-defined function for thermal conductivity that is pressure dependent in the form of

$$k(p) = 0{,}045809(\log 10(p)) + 0{,}006317$$

where k is the thermal conductivity in W/(m·K) and p is the pressure in Torr. This user-defined function is specified in the material model for helium using a piecewise linear function.

BOUNDARY CONDITIONS

Laminar Flow

A **Laminar Flow** interface is applied to the region between the wafer and the dielectric blocks. An **Inlet** boundary is applied with mass flow in standard flow rate (SCCM) and a mean molar mass of 0.004 kg/mol for helium gas.

Solid Mechanics

Two **Contact** boundaries are applied at the contact points between the wafer and the e-chuck. The first contact boundary is at the edge of the wafer where it is in contact with the e-chuck. The second contact boundary is in the middle of the wafer to constrain the displacement of the wafer when pull-in occurs. **Gravity** (a volume force) is applied to the wafer. **Solid Mechanics** is only applied to the wafer because the dielectric blocks of the e-chuck can be assumed rigid.

Heat Transfer in Solids and Fluids

This model applies **Initial Values** to the wafer as the initial condition in the time-dependent simulation. Time-dependent **Heat Flux** is applied to the top surface of the wafer to represent a heat pulse from a plasma process. The **Heat Transfer** interface is applied to the entire geometry.

Electrostatics

The **Electrostatics** interface is applied to the entire geometry. The voltage required to hold the wafer in place depends on the pressure from the helium flow. In this model, an **Integration Operator** is used to access the z-coordinate of `p17`. Then, the **Global Equation** `intop1(z)-zsp=0` can be solved for the value of `VdcSP` needed to hold `p17` at the setpoint `zsp`. **Terminal** boundaries are applied to the bottom of the wafer and the dielectric blocks.

Deforming Domain

The mesh in the rectangular region between the wafer and the e-chuck is set to deform freely, following a hyperelastic mesh smoothing deformation. The mesh displacement is controlled by the structural displacement of the wafer boundaries. At the dielectric blocks boundaries, a fixed boundary is used.

STUDIES AND STRATEGY FOR CONVERGENCE

This highly nonlinear problem requires ramping of a parameter such as the helium flow rate for it to converge to a solution. The studies should be done in steps with an increasing number of coupled interfaces. The solution of a previous study can be used as the initial value by a subsequent study:

- Study 1: Only **Solid Mechanics** and **Electrostatics** are enabled to study voltage conditions without gas flow. The study computes the wafer's profile when `zsp` is set to 35, 30, 25, 20, and 15 μm. When `zsp` = 15 μm, the center of the wafer is in contact with the e-chuck.
- Study 2: The **Solid Mechanics**, **Electrostatics**, and **Laminar Flow** interfaces are enabled to study voltage conditions when the mass flow rate is set to 0, 100, 200, 300, and 400 SCCM and `zsp` = 30 μm. This study computes the wafer profile and the pressure field within the fluid region.
- Study 3: In this time-dependent study, all four interfaces, including **Heat Transfer**, are enabled. This study uses the solution of Study 2 as the initial solution to solve for the temperature field resulting from heat transfer between the e-chuck and the wafer and the plasma process.

Moreover, the use of Global Equations and the coupled physics necessitates adjustments to the solver settings. In this model, the **Fully Coupled** solver with Automatic highly nonlinear Newton method is required instead of the default **Segregated** solver.

Results and Discussion

In **Study 1**, `p17` is set to 35, 30, 25, 20, and 15 um without helium gas flowing. The resulting wafer profile is shown in Figure 2. `VdcSP` versus z-displacement at `p17` is shown in Figure 3. In **Study 2**, `zsp` is set to 30 μm and the helium gas flow = 0, 100, 200, 300, and 400 SSCM. The resulting wafer profile is shown in Figure 4 and the plot of `VdcSP` versus helium flow rate is shown in Figure 5. The surface plot of the displacement and the deformation is shown in Figure 6. In the time-dependent Study 3, a heat pulse is introduced, resulting in the plot of wafer temperature versus time shown in Figure 7.

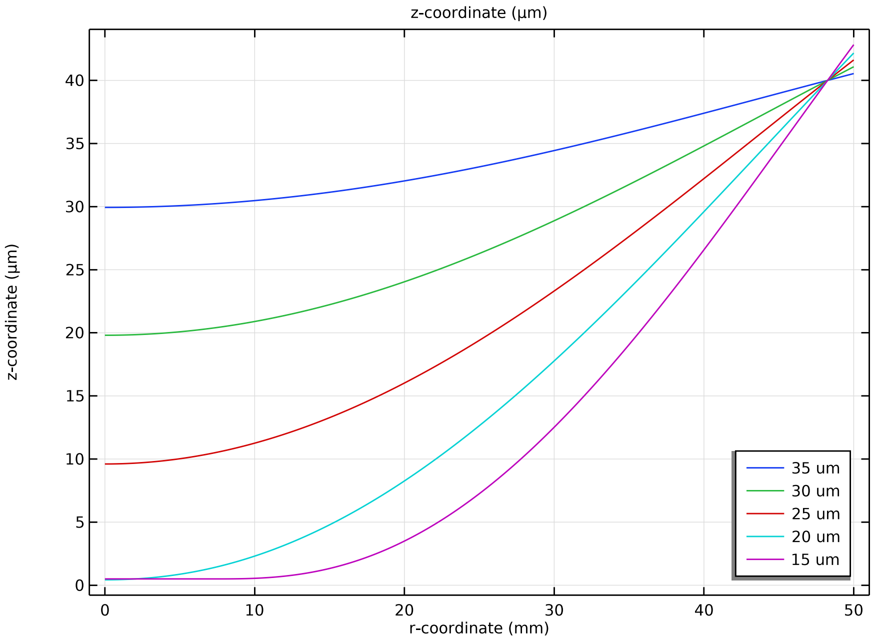


Figure 2: Wafer profile when `zsp` *is set to 35, 30, 25, 20, and 15* µm *without helium gas flowing. As expected. The structural displacement is maximal at the center of the geometry. For zsp = 15 and 20* µm*, the center of the wafer is in contact with the e-chuck.*

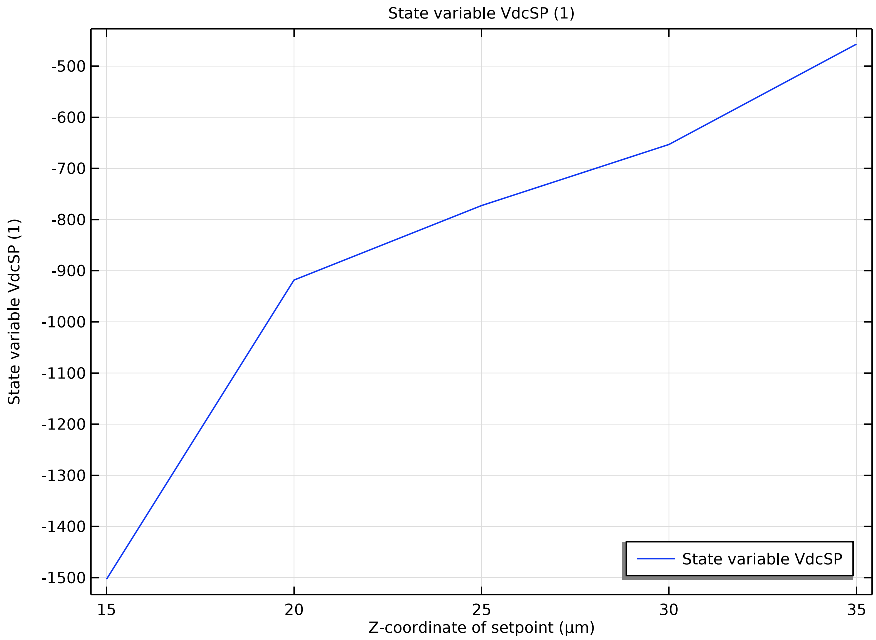


Figure 3: `VdcSP` *versus* `zsp`*, without helium gas flowing.*

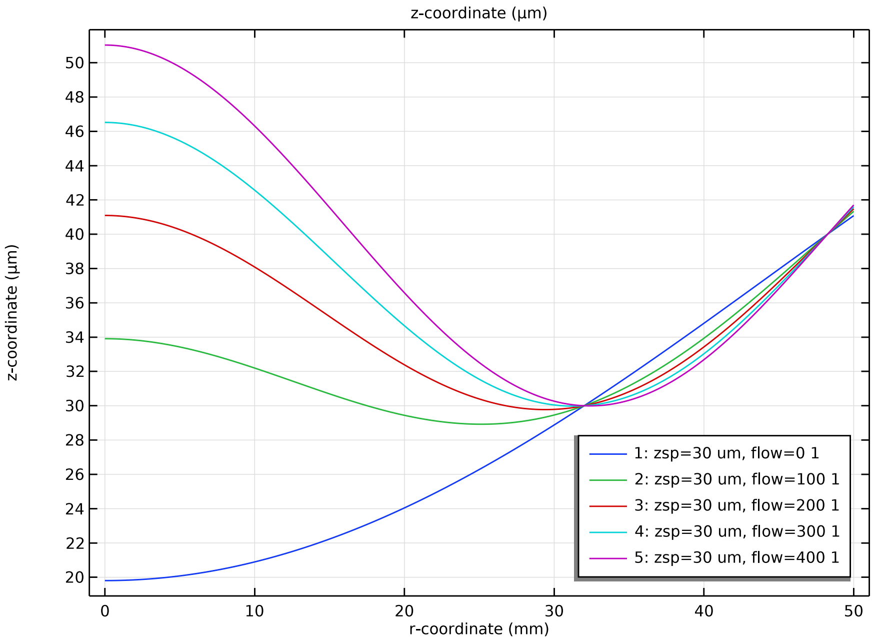


Figure 4: Wafer profile when `zsp` *= 30 µm with helium gas flow = 0, 100, 200, 300, and 400 SCCM.*

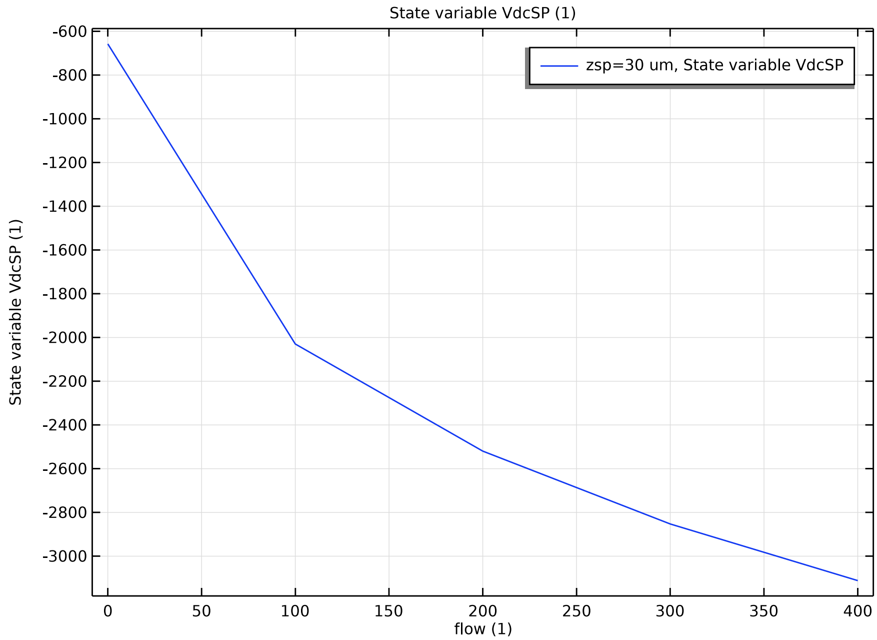


Figure 5: VdcSP versus helium gas flow for `zsp` *= 30 µm. Helium gas flow = 0, 100, 200, 300, or 400 SCCM.*

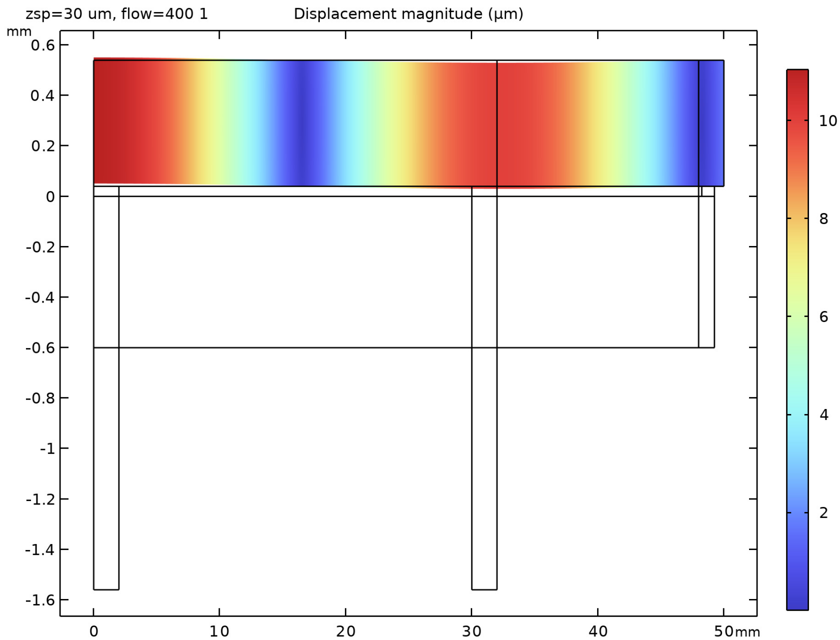


Figure 6: Surface Plot and Deformation for `zsp` *= 30 µm and helium gas flow = 400 SCCM.*

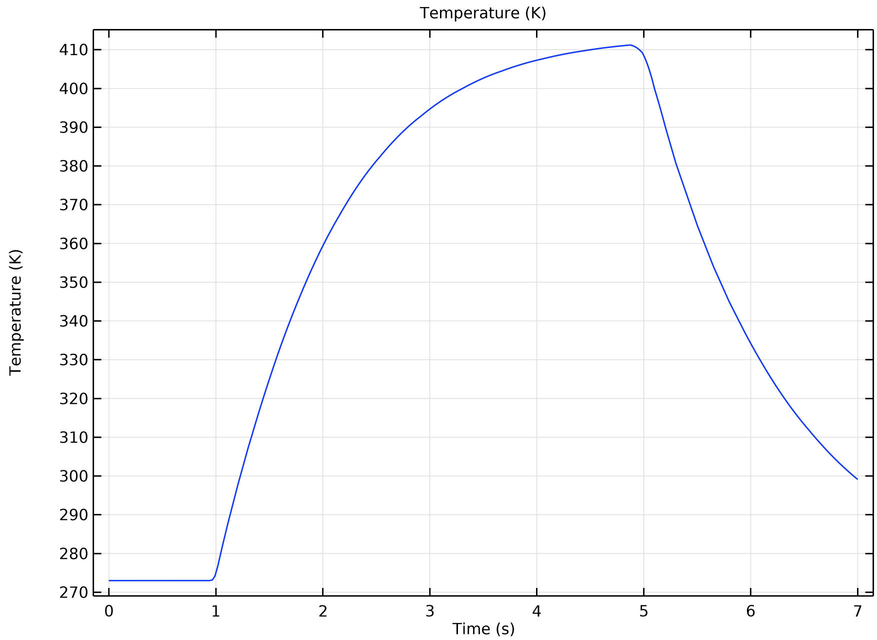


Figure 7: Time-dependent surface temperature at the center of the wafer with helium gas flow of 100 SCCM.

Application Library path: `MEMS_Module/Actuators/electrostatic_chuck`

Modeling Instructions

Start by creating a new 2D axisymmetric model with (i) **Fluid-Solid Interaction** (multiphysics interface), (ii) **Electrostatics** (physics), and (iii) **Electromechanics** (multiphysics coupling). Built-in to the **Fluid-Solid Interaction** are **Laminar Flow**, **Solid Mechanics**, and **Heat Transfer in Solids and Fluids**.

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

NEW

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

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.
2 In the **Select Physics** tree, select **Fluid Flow > Fluid–Structure Interaction > Conjugate Heat Transfer > Fluid–Solid Interaction**.
3 Click **Add**.
4 In the **Select Physics** tree, select **AC/DC > Electric Fields and Currents > Electrostatics (es)**.
5 Click **Add**.
6 Click **Done**.

MULTIPHYSICS

Electromechanics, Solid 1 (eme1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain > Electromechanics, Solid**.

Import parameters for Geometry, Mesh, and Process for convenience.

GLOBAL DEFINITIONS

Geometry

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, type `Geometry` in the **Label** text field.
3 Locate the **Parameters** section. Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `electrostatic_chuck_geometry_parameters.txt`.

Mesh

1 In the **Home** toolbar, click **Parameters** and choose **Add > Parameters**.
2 In the **Settings** window for **Parameters**, type `Mesh` in the **Label** text field.
3 Locate the **Parameters** section. Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `electrostatic_chuck_mesh_parameters.txt`.

Process

1 In the **Home** toolbar, click **Parameters** and choose **Add > Parameters**.
2 In the **Settings** window for **Parameters**, type `Process` in the **Label** text field.
3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `electrostatic_chuck_process_parameters.txt`.

DEFINITIONS

Rectangle 1 (rect1)

1 In the **Definitions** toolbar, click f(x) **More Functions** and choose **Rectangle**.

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

3 In the **Lower limit** text field, type `t_start`.

4 In the **Upper limit** text field, type `t_start+t_pulse`.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Qt	Q_plasma*rect1(t)	W/m²	

Set the geometry unit to millimeters for convenience.

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**.

The geometry is comprised of wafer, insulator, gap, inlet, and outlet.

Wafer

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

2 In the **Settings** window for **Rectangle**, type `Wafer` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `d_wafer/2`.

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

5 Locate the **Position** section. In the **z** text field, type `gap`.

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog, type `Selection: Wafer` in the **Name** text field.

8 Click **OK**.

Gap 1

1. In the **Geometry** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, type `Gap 1` in the **Label** text field.
3. Locate the **Size and Shape** section. In the **Width** text field, type `x_outlet`.
4. In the **Height** text field, type `gap`.

Gap 2

1. In the **Geometry** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, type `Gap 2` in the **Label** text field.
3. Locate the **Size and Shape** section. In the **Width** text field, type `x_gap3-x_outlet`.
4. In the **Height** text field, type `gap`.
5. Locate the **Position** section. In the **r** text field, type `x_outlet`.

Gap 3

1. In the **Geometry** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, type `Gap 3` in the **Label** text field.
3. Locate the **Size and Shape** section. In the **Width** text field, type `w_gap3`.
4. In the **Height** text field, type `gap`.
5. Locate the **Position** section. In the **r** text field, type `x_gap3`.

Pin

1. In the **Geometry** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, type `Pin` in the **Label** text field.
3. Locate the **Size and Shape** section. In the **Width** text field, type `d_pin`.
4. In the **Height** text field, type `gap`.
5. Locate the **Position** section. In the **r** text field, type `x_gap3+w_gap3`.
6. Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.
7. In the **New Cumulative Selection** dialog, type `Insulator` in the **Name** text field.
8. Click **OK**.

Rectangle 6 (r6)

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

GEOMETRY 1

Insulator 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** > **View 1** node, then click **Component 1 (comp1)** > **Geometry 1** > **Rectangle 6 (r6)**.
2 In the **Settings** window for **Rectangle**, type `Insulator 1` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Width** text field, type `x_outlet-d_inlet/2`.
4 In the **Height** text field, type `t_insulator`.
5 Locate the **Position** section. In the **r** text field, type `d_inlet/2`.
6 In the **z** text field, type `-t_insulator`.
7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Insulator**.

Insulator 2

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, type `Insulator 2` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Width** text field, type `x_gap3+w_gap3+d_pin-x_outlet-w_outlet`.
4 In the **Height** text field, type `t_insulator`.
5 Locate the **Position** section. In the **r** text field, type `x_outlet+w_outlet`.
6 In the **z** text field, type `-t_insulator`.
7 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Insulator**.

Inlet

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, type `Inlet` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Width** text field, type `d_inlet/2`.
4 In the **Height** text field, type `L_inlet+t_insulator`.
5 Locate the **Position** section. In the **z** text field, type `-L_inlet-t_insulator+gap`.

Outlet

1 In the **Geometry** toolbar, click **Rectangle**.
2 In the **Settings** window for **Rectangle**, type `Outlet` in the **Label** text field.
3 Locate the **Size and Shape** section. In the **Width** text field, type `w_outlet`.
4 In the **Height** text field, type `L_outlet+t_insulator`.

5 Locate the **Position** section. In the **r** text field, type `x_outlet`.

6 In the **z** text field, type `-L_inlet-t_insulator+gap`.

Union 1 (uni1)

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

2 Select the objects **r1**, **r2**, **r3**, **r4**, **r8**, and **r9** only.

Union 2 (uni2)

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

2 Select the objects **r5**, **r6**, and **r7** only.

Partition the domains so that two contact regions can be created.

Partition Domains 1 (pard1)

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

2 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

3 Click to select the **Activate Selection** toggle button for **Vertices defining line segments**.

4 From the **Partition with** list, choose **Extended edges**.

5 Click to select the **Activate Selection** toggle button for **Domains to partition**.

6 Click to select the **Activate Selection** toggle button for **Straight or circular edges**.

7 On the object **uni2**, select Boundary 2 only.

8 Click to select the **Activate Selection** toggle button for **Domains to partition**.

9 On the object **uni1**, select Domain 1 only.

Partition Domains 2 (pard2)

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

2 On the object **pard1**, select Domain 6 only.

3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.

4 Click to select the **Activate Selection** toggle button for **Vertices defining line segments**.

5 Click in the **Graphics** window and then press Ctrl+D to clear all objects.

6 From the **Partition with** list, choose **Extended edges**.

7 On the object **uni2**, select Boundary 2 only.

Partition Domains 3 (pard3)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.
2 On the object **uni2**, select Domain 2 only.
3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.
4 Click to select the **Activate Selection** toggle button for **Vertices defining line segments**.
5 From the **Partition with** list, choose **Extended edges**.
6 On the object **pard2**, select Boundary 27 only.

Partition Domains 4 (pard4)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Domains**.
2 On the object **pard2**, select Domain 4 only.
3 In the **Settings** window for **Partition Domains**, locate the **Partition Domains** section.
4 Click to select the **Activate Selection** toggle button for **Vertices defining line segments**.
5 From the **Partition with** list, choose **Extended edges**.
6 On the object **pard3**, select Boundaries 5 and 8 only.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Form Union (fin)**.
2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.
3 From the **Action** list, choose **Form an assembly**. This is necessary because the model includes moving objects.
4 Clear the **Create pairs** checkbox.
5 Click **Build Selected**.

DEFINITIONS

Axis

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Definitions** > **View 1** click **Axis**.
2 In the **Settings** window for **Axis**, locate the **Axis** section.
3 From the **View scale** list, choose **Manual**.

4 In the **y scale** text field, type 20.

5 Click **Update**.

View 1

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

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

3 Select the **Show geometry labels** checkbox.

Add a nonlocal integration coupling to compute the actual displacement.

Integration 1 (intop1)

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

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 17 in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Integration**, locate the **Advanced** section.

8 Clear the **Compute integral in revolved geometry** checkbox.

Add a **Contact Pair** and an **Identity Boundary Pair**.

Contact Pair 1 (p1)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 47 49 51 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

7 Click to select the **Activate Selection** toggle button.

8 Click **Paste Selection**.

9 In the **Paste Selection** dialog, type 32 35 in the **Selection** text field.

10 Click **OK**.

11 In the **Settings** window for **Pair**, locate the **Advanced** section.

12 From the **Mapping method** list, choose **Initial configuration**.

Contact Pair 2 (p2)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `37 39 40` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

7 Click to select the **Activate Selection** toggle button.

8 Click **Paste Selection**.

9 In the **Paste Selection** dialog, type `8 14 21` in the **Selection** text field.

10 Click **OK**.

11 In the **Settings** window for **Pair**, locate the **Advanced** section.

12 From the **Mapping method** list, choose **Initial configuration**.

Identity Boundary Pair 3 (p3)

1 In the **Definitions** toolbar, click **Pairs** and choose **Identity Boundary Pair**.

2 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `37 39 40 41 43 46 47 49` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

7 Click to select the **Activate Selection** toggle button.

8 Click **Paste Selection**.

9 In the **Paste Selection** dialog, type `13 25 30 34 35` in the **Selection** text field.

10 Click **OK**.

Add an explicit selection for the fluid domain.

Fluid

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Fluid` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `1 2 3 5 6 7 8 9 11` in the **Selection** text field.

5 Click **OK**.

Solid

1. In the **Definitions** toolbar, click **Complement**.
2. In the **Settings** window for **Complement**, type `Solid` in the **Label** text field.
3. Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.
4. In the **Add** dialog, select **Fluid** in the **Selections to invert** list.
5. Click **OK**.

MOVING MESH

Deforming Domain 1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Moving Mesh** click **Deforming Domain 1**.
2. In the **Settings** window for **Deforming Domain**, locate the **Domain Selection** section.
3. Click **Paste Selection**.
4. In the **Paste Selection** dialog, type `1 2 3 5 6 7 8 9 11` in the **Selection** text field.
5. Click **OK**.
6. In the **Settings** window for **Deforming Domain**, locate the **Smoothing** section.
7. From the **Mesh smoothing type** list, choose **Hyperelastic**.

Add a **Symmetry/Roller** boundary on the axis of symmetry bordering the **Deforming Domain**.

Symmetry/Roller 1

1. In the **Moving Mesh** toolbar, click **Symmetry/Roller**.
2. Select Boundaries 1, 3, and 5 only.

Add materials to the model.

ADD MATERIAL

1. In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **MEMS** > **Insulators** > **Al2O3 - Aluminum oxide**.
4. Click the **Add to Component** button in the window toolbar.

MATERIALS

Al2O3 - Aluminum oxide (mat1)

1. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2. From the **Selection** list, choose **Insulator**.

ADD MATERIAL

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

2 In the tree, select **MEMS** > **Semiconductors** > **Si - Silicon (single-crystal, isotropic)**.

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

MATERIALS

Si - Silicon (single-crystal, isotropic) (mat2)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Selection: Wafer**.

ADD MATERIAL

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

2 In the tree, select **Liquids and Gases** > **Gases** > **Helium**.

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

4 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Helium (mat3)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Fluid**.

3 In the **Model Builder** window, expand the **Helium (mat3)** node.

Define a piecewise continuous function for the pressure-dependent thermal conductivity of the gas.

Piecewise 3 (k)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Materials** > **Helium (mat3)** > **Basic (def)** node, then click **Piecewise 3 (k)**.

2 In the **Settings** window for **Piecewise**, locate the **Definition** section.

3 In the **Argument** text field, type `pA`.

4 Find the **Intervals** subsection. In the table, enter the following settings:

Start	End	Function
1	100	`0.045809*log10(pA)+0.0063167`

5 Locate the **Units** section. In the **Arguments** text field, type `Torr`.

Set up **Laminar Flow** boundary conditions.

LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Fluid**.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Laminar Flow (spf)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the p text field, type `0.1`.

Flow Continuity 1

1 In the **Model Builder** window, click **Flow Continuity 1**.

2 In the **Settings** window for **Flow Continuity**, locate the **Advanced** section.

3 Select the **Disconnect pair** checkbox.

Specify an **Inlet** boundary condition with Mass Flow in Standard flow rate (SCCM).

Inlet 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `2` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

7 From the list, choose **Mass flow**.

8 Locate the **Mass Flow** section. From the **Mass flow type** list, choose **Standard flow rate (SCCM)**.

9 In the Q_{sccm} text field, type `flow`.

10 In the M_n text field, type `mass_He`.

Outlet 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `16` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Outlet**, locate the **Pressure Conditions** section.

7 In the p_0 text field, type `0.01[Torr]`.

8 Clear the **Suppress backflow** checkbox.

Set up **Solid Mechanics** boundary conditions.

With the assumption that the electrostatic chuck is rigid and only the silicon wafer will be deformed, only the silicon domain is selected for the **Solid Mechanics** interface. This reduces the computation load as the solid-mechanics degrees of freedom outside of the silicon domain are not solved for.

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Selection: Wafer**.

Gravity 1

In the **Physics** toolbar, click **Global** and choose **Gravity**.

Contact 1a

1 In the **Physics** toolbar, click **Pairs** and choose **Contact**.

Set up the contact condition between the wafer and the chuck surface.

2 In the **Settings** window for **Contact**, locate the **Contact Pressure Penalty Factor** section.

3 From the **Penalty factor control** list, choose **Manual tuning**.

4 In the f_p text field, type `1/10`.

5 Click to expand the **Contact Surface Offset and Adjustment** section. In the $d_{\mathrm{offset,d}}$ text field, type `0.5[um]`.

6 Locate the **Pair Selection** section. Click **Add**.

7 In the **Add** dialog, select **Contact Pair 2 (p2)** in the **Pairs** list.

8 Click **OK**.

Set up the contact condition between the wafer and the pin.

Contact 1

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

2 In the **Settings** window for **Contact**, locate the **Contact Pressure Penalty Factor** section.

3 From the **Penalty factor control** list, choose **Manual tuning**.

4 In the f_{p} text field, type `1/10`.

5 Locate the **Contact Surface Offset and Adjustment** section. Select the **Force zero initial gap** checkbox.

Set up the **Heat Transfer in Solids and Fluids** boundary conditions.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids and Fluids (ht)**.

2 In the **Settings** window for **Heat Transfer in Solids and Fluids**, locate the **Domain Selection** section.

3 In the list box, select **16**.

4 Click **Remove from Selection**.

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

6 Locate the **Physical Model** section. In the T_{ref} text field, type `T_chuck`.

Solid 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Heat Transfer in Solids and Fluids (ht)** click **Solid 1**.

2 In the **Settings** window for **Solid**, locate the **Model Input** section.

3 From the T_{ref} list, choose **User defined**. In the associated text field, type `T_chuck`.

4 From the p_A list, choose **Common model input**.

Fluid 1

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

2 In the **Settings** window for **Fluid**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Fluid**.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T text field, type `T_chuck`.

Solid 2 - Wafer

1 In the **Physics** toolbar, click **Domains** and choose **Solid**.

2 In the **Settings** window for **Solid**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Selection: Wafer**.

4 In the **Label** text field, type `Solid 2 - Wafer`.

5 Locate the **Model Input** section. From the T_{ref} list, choose **User defined**. In the associated text field, type `T_chuck`.

6 From the p_A list, choose **Absolute pressure (spf)**.

Temperature 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `10 15 22 30 38 42 45 46 60` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Temperature**, locate the **Temperature** section.

7 In the T_0 text field, type `T_chuck`.

Thermal Insulation 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Thermal Insulation**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `46 48` in the **Selection** text field.

5 Click **OK**.

Specify a **Heat Flux** boundary condition along the top surface of the wafer to model the plasma process.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `9 28 33` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

7 In the q_0 text field, type `Qt`.

Initial Values 2

1. In the **Physics** toolbar, click **Domains** and choose **Initial Values**.
2. In the **Settings** window for **Initial Values**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Selection: Wafer**.
4. Locate the **Initial Values** section. In the T text field, type `T_wafer_init`.

Set up **Electromechanics** boundary conditions.

ELECTROSTATICS (ES)

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

Charge Conservation in Solids 1

1. In the **Physics** toolbar, click **Domains** and choose **Charge Conservation in Solids**.
2. In the **Settings** window for **Charge Conservation in Solids**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **Solid**.

Electric Continuity 1

1. In the **Physics** toolbar, click **Pairs** and choose **Electric Continuity**.
2. In the **Settings** window for **Electric Continuity**, locate the **Pair Selection** section.
3. Click **Add**.
4. In the **Add** dialog, select **Identity Boundary Pair 3 (p3)** in the **Pairs** list.
5. Click **OK**.

Boundary Terminal 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Boundary Terminal**.
2. In the **Settings** window for **Boundary Terminal**, locate the **Boundary Selection** section.
3. Click **Paste Selection**.
4. In the **Paste Selection** dialog, type `8 14 21 27 32 35` in the **Selection** text field.
5. Click **OK**.
6. In the **Settings** window for **Boundary Terminal**, locate the **Terminal** section.
7. From the **Terminal type** list, choose **Voltage**.
8. In the V_0 text field, type `0`.

Change the drive potential to the value `VdcSP`, which will be solved for in a global equation.

Boundary Terminal 2

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `38 42` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Boundary Terminal**, locate the **Terminal** section.

7 From the **Terminal type** list, choose **Voltage**.

8 In the V_0 text field, type `VdcSP[V]`.

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

10 In the **Show More Options** dialog, in the tree, select the checkbox for the node **Physics** > **Equation Contributions**.

11 Click **OK**.

Add a **Global Equation** to compute the voltage for a given displacement, `VdcSP`.

Global Equations 1 (ODE1)

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 (ut_0) (1/s)	Description
VdcSP	(intop1(z)-zsp)/zsp	0	0	

To define the structured mesh needed by the contact boundary conditions, several steps are required.

MESH 1

Edge 1

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `2 4 6 8` in the **Selection** text field.

5 Click **OK**.

Distribution 1

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

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

3 In the **Number of elements** text field, type `10*(d_inlet/2)`.

Edge 2

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `16 18 20 21` in the **Selection** text field.

5 Click **OK**.

Distribution 1

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

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

3 In the **Number of elements** text field, type `20*(w_outlet/2)`.

Edge 3

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `1 3 10 11 15 17 22 23` in the **Selection** text field.

5 Click **OK**.

Distribution 1

1 Right-click **Edge 3** and choose **Distribution**.

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

3 From the **Distribution type** list, choose **Predefined**.

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

5 In the **Element ratio** text field, type `50`.

6 Select the **Reverse direction** checkbox.

Edge 4

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 37 40 41 44 50 in the **Selection** text field.

5 Click **OK**.

Distribution 1

1 Right-click **Edge 4** and choose **Distribution**.

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

3 From the **Distribution type** list, choose **Predefined**.

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

5 In the **Element ratio** text field, type 2.

6 Select the **Reverse direction** checkbox.

Mapped 1 - Inlet and Outlet

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, type `Mapped 1 - Inlet and Outlet` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Geometric entity level** list, choose **Domain**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 1 2 6 7 in the **Selection** text field.

6 Click **OK**.

Edge 5

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type 5 12 19 24 29 34 47 51 in the **Selection** text field.

5 Click **OK**.

Distribution 1

1 Right-click **Edge 5** and choose **Distribution**.

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

3 From the **Distribution type** list, choose **Predefined**.

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

5 In the **Element ratio** text field, type 5.

6 Select the **Symmetric distribution** checkbox.

Mapped 2 - Corner

1 In the **Mesh** toolbar, click **Mapped**.

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `3 8` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Mapped**, type `Mapped 2 - Corner` in the **Label** text field.

Edge 6

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `13 25 38 39 42 43` in the **Selection** text field.

5 Click **OK**.

Distribution 1

1 Right-click **Edge 6** and choose **Distribution**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `13` in the **Selection** text field.

6 Click **OK**.

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

8 In the **Number of elements** text field, type `mfx*x_outlet`.

Distribution 2

1 In the **Model Builder** window, right-click **Edge 6** and choose **Distribution**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `25` in the **Selection** text field.

6 Click **OK**.

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

8 In the **Number of elements** text field, type `mfx*(d_wafer/2-x_outlet)`.

Distribution 3

1 Right-click **Edge 6** and choose **Distribution**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `38 39` in the **Selection** text field.

6 Click **OK**.

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

8 In the **Number of elements** text field, type `2*x_outlet`.

Distribution 4

1 Right-click **Edge 6** and choose **Distribution**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `42 43` in the **Selection** text field.

6 Click **OK**.

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

8 In the **Number of elements** text field, type `2*(d_wafer/2-x_outlet)`.

Mapped - Channel

1 In the **Mesh** toolbar, click **Mapped**.

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

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `5 9` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Mapped**, type `Mapped - Channel` in the **Label** text field.

Mapped - Insulator

1 In the **Mesh** toolbar, click **Mapped**.

2 In the **Settings** window for **Mapped**, type `Mapped - Insulator` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Geometric entity level** list, choose **Domain**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `13 14` in the **Selection** text field.

6 Click **OK**.

Edge 7

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

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

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `30 32 49` in the **Selection** text field.

5 Click **OK**.

Distribution 1

1 Right-click **Edge 7** and choose **Distribution**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `49` in the **Selection** text field.

6 Click **OK**.

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

8 From the **Distribution type** list, choose **Predefined**.

9 In the **Number of elements** text field, type `my_channel`.

Distribution 2

1 In the **Model Builder** window, right-click **Edge 7** and choose **Distribution**.

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

3 Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type `30 32` in the **Selection** text field.

6 Click **OK**.

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

8 From the **Distribution type** list, choose **Predefined**.

9 In the **Number of elements** text field, type `3*mx_pin`.

10 In the **Element ratio** text field, type `2`.

Mapped 5

1 In the **Mesh** toolbar, click **Mapped**.
2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog, type 11 in the **Selection** text field.
6 Click **OK**.

Edge 8

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.
2 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog, type 46 in the **Selection** text field.
5 Click **OK**.

Distribution 1

1 Right-click **Edge 8** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.
3 Click **Clear Selection**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog, type 46 in the **Selection** text field.
6 Click **OK**.
7 In the **Settings** window for **Distribution**, locate the **Distribution** section.
8 From the **Distribution type** list, choose **Predefined**.
9 In the **Number of elements** text field, type 2.
10 Select the **Symmetric distribution** checkbox.

Mapped 6

1 In the **Mesh** toolbar, click **Mapped**.
2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog, type 16 in the **Selection** text field.
6 Click **OK**.

Edge 9

1 In the **Mesh** toolbar, click **More Generators** and choose **Edge**.
2 In the **Settings** window for **Edge**, locate the **Boundary Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog, type `7 36` in the **Selection** text field.
5 Click **OK**.

Distribution 1

1 Right-click **Edge 9** and choose **Distribution**.
2 In the **Settings** window for **Distribution**, locate the **Distribution** section.
3 In the **Number of elements** text field, type `my_wafer`.

Copy Edge 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations** > **Copy Edge**.
2 Select Boundaries 46 and 48 only.
3 In the **Settings** window for **Copy Edge**, locate the **Destination Boundaries** section.
4 Click to select the **Activate Selection** toggle button.
5 Select Boundary 45 only.

Mapped 7

1 In the **Mesh** toolbar, click **Mapped**.
2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog, type `15` in the **Selection** text field.
6 Click **OK**.

Free Triangular 1

1 In the **Mesh** toolbar, click **Free Triangular**.
2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Click **Paste Selection**.
5 In the **Paste Selection** dialog, type `4 10 12` in the **Selection** text field.
6 Click **OK**.

7 In the **Settings** window for **Free Triangular**, click **Build All**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

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

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

Set up a **Stationary** study with a parametric sweep over the displacement set point, `zsp`, without gas flow.

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 the **Add Study** button in the window toolbar.

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

STUDY 1 - WITHOUT GAS FLOW

1 In the **Settings** window for **Study**, type `Study 1 - Without Gas Flow` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1 - Without Gas Flow** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Laminar Flow (spf)**.

4 In the **Solve for** column of the table, under **Component 1 (comp1)**, select the checkbox for **Solid Mechanics (solid)**.

5 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Heat Transfer in Solids and Fluids (ht)**.

6 In the **Solve for** column of the table, under **Component 1 (comp1)** > **Multiphysics**, clear the checkboxes for **Fluid–Structure Interaction 1 (fsi1)**, **Nonisothermal Flow 1 (nitf1)**, and **Thermal Expansion 1 (te1)**.

7 Click to expand the **Values of Dependent Variables** section. Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

8 Click **Add**.

9 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
zsp (Z-coordinate of setpoint)	`35 30 25 20 15`	`um`

The problem is highly nonlinear due to coupled physics and the presence of the global equation, so the solver settings need to be adjusted accordingly.

Solution 1 (sol1)

1. In the **Study** toolbar, click **Show Default Solver**.
2. In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
3. In the **Model Builder** window, expand the **Study 1 - Without Gas Flow** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** node.
4. Right-click **Study 1 - Without Gas Flow** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** and choose **Fully Coupled**.
5. In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
6. From the **Nonlinear method** list, choose **Automatic highly nonlinear (Newton)**.
7. In the **Maximum number of iterations** text field, type `200`.
8. In the **Study** toolbar, click **Compute**.

Set up a **Stationary** study with a parametric sweep over the displacement set point, `zsp`, with gas flow.

ADD STUDY

1. In the **Study** 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 the **Add Study** button in the window toolbar.
5. In the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2 - WITH GAS FLOW

1. In the **Settings** window for **Study**, type `Study 2 - With Gas Flow` in the **Label** text field.
2. Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Use the solution of the previous study as initial values of variables solved for.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2 - With Gas Flow** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Heat Transfer in Solids and Fluids (ht)**.

4 In the **Solve for** column of the table, under **Component 1 (comp1)** > **Multiphysics**, clear the checkbox for **Thermal Expansion 1 (te1)**.

5 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

6 From the **Method** list, choose **Solution**.

7 From the **Study** list, choose **Study 1 - Without Gas Flow, Stationary**.

8 From the **Parameter value (zsp (um))** list, choose **30 um**.

9 Locate the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

10 From the **Sweep type** list, choose **All combinations**.

11 Click **Add**.

12 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
flow (Mass flow in SCCM)	range(0,100,400)	1

13 Click **Add**.

14 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
zsp (Z-coordinate of setpoint)	30	um

The problem is highly nonlinear due to coupled physics and the presence of the global equation, so the solver settings need to be adjusted accordingly.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2 - With Gas Flow** > **Solver Configurations** > **Solution 2 (sol2)** > **Stationary Solver 1** node.

4 Right-click **Study 2 - With Gas Flow** > **Solver Configurations** > **Solution 2 (sol2)** > **Stationary Solver 1** and choose **Fully Coupled**.

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

Set up a **Time Dependent** study with a **Heat Flux** boundary condition on the wafer surface to model heating from the plasma.

ADD STUDY

1 In the **Study** 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** > **Time Dependent**.

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

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

STUDY 3 - WAFER TEMPERATURE VS. TIME

1 In the **Settings** window for **Study**, type `Study 3 - Wafer Temperature vs. Time` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Use the solution of the previous study as initial values of variables solved for.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 3 - Wafer Temperature vs. Time** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 In the **Solve for** column of the table, under **Component 1 (comp1)** > **Multiphysics**, clear the checkbox for **Thermal Expansion 1 (te1)**.

4 Locate the **Study Settings** section. From the **Time unit** list, choose **ms**.

5 In the **Output times** text field, type `range(0,5,7000)`.

6 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

7 From the **Method** list, choose **Solution**.

8 From the **Study** list, choose **Study 2 - With Gas Flow, Stationary**.

9 From the **Parameter value (zsp (um),flow (l))** list, choose **2: zsp=30 um, flow=100 l**.

10 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

11 Click **Add**.

12 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
flow (Mass flow in SCCM)	100	1

13 Click **Add**.

14 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
zsp (Z-coordinate of setpoint)	30	um

15 Click **Add**.

16 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Q_plasma (Plasma heat flux)	1E5	W/m^2

The problem is highly nonlinear due to coupled physics and the presence of the global equation, so the solver settings need to be adjusted accordingly.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

3 In the **Model Builder** window, expand the **Study 3 - Wafer Temperature vs. Time** > **Solver Configurations** > **Solution 3 (sol3)** > **Time-Dependent Solver 1** node.

4 Right-click **Study 3 - Wafer Temperature vs. Time** > **Solver Configurations** > **Solution 3 (sol3)** > **Time-Dependent Solver 1** and choose **Fully Coupled**.

5 In the **Settings** window for **Fully Coupled**, locate the **General** section.

6 From the **Linear solver** list, choose **Direct, spatial mesh displacement (spf)**.

7 Click to expand the **Method and Termination** section. From the **Nonlinear method** list, choose **Automatic highly nonlinear (Newton)**.

8 In the **Maximum number of iterations** text field, type 200.

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

From the results of Study 1, plot the wafer profile.

RESULTS

Wafer Profile, Without Gas Flow

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Wafer Profile, Without Gas Flow` in the **Label** text field.

Line Graph 1

1 Right-click **Wafer Profile, Without Gas Flow** and choose **Line Graph**.
2 In the **Settings** window for **Line Graph**, locate the **Selection** section.
3 Click **Paste Selection**.
4 In the **Paste Selection** dialog, type `8 14 21 27 32 35` in the **Selection** text field.
5 Click **OK**.
6 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
7 In the **Expression** text field, type `z`.
8 From the **Unit** list, choose **µm**.
9 Select the **Description** checkbox.
10 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
11 In the **Expression** text field, type `r`.
12 Click to expand the **Legends** section. Select the **Show legends** checkbox.

Wafer Profile, Without Gas Flow

1 In the **Model Builder** window, click **Wafer Profile, Without Gas Flow**.
2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.
3 From the **Position** list, choose **Lower right**.
4 In the **Wafer Profile, Without Gas Flow** toolbar, click **Plot**.

From the results of Study 1, plot `VdcSP` versus *z* setpoints.

VdcSP vs. zsp

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `VdcSP vs. zsp` in the **Label** text field.

Global 1

1 Right-click **VdcSP vs. zsp** 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**
`VdcSP`	`1`	`State variable VdcSP`

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

5 In the **Expression** text field, type `zsp`.

6 From the **Unit** list, choose **µm**.

7 Select the **Description** checkbox.

8 In the **VdcSP vs. zsp** toolbar, click **Plot**.

From the results of Study 2, plot the wafer profile.

Wafer Profile, With Gas Flow

1 In the **Model Builder** window, right-click **Wafer Profile, Without Gas Flow** and choose **Duplicate**.

2 In the **Model Builder** window, click **Wafer Profile, Without Gas Flow 1**.

3 In the **Settings** window for **1D Plot Group**, type `Wafer Profile, With Gas Flow` in the **Label** text field.

4 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - With Gas Flow/ Solution 2 (sol2)**.

5 In the **Wafer Profile, With Gas Flow** toolbar, click **Plot**.

6 Click **Plot**.

From the results of Study 2, plot `VdcSP` versus the gas-flow rate.

VdcSP vs. zsp

1 In the **Model Builder** window, click **VdcSP vs. zsp**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Lower right**.

4 In the **VdcSP vs. zsp** toolbar, click **Plot**.

VdcSP vs. Gas Flow

1 Right-click **VdcSP vs. zsp** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `VdcSP vs. Gas Flow` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - With Gas Flow/ Solution 2 (sol2)**.

Global 1

1 In the **Model Builder** window, expand the **VdcSP vs. Gas Flow** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **x-Axis Data** section.

3 From the **Axis source data** list, choose **flow**.

4 From the **Parameter** list, choose **Parameter value**.

5 In the **VdcSP vs. Gas Flow** toolbar, click **Plot**.

VdcSP vs. Gas Flow

1 In the **Model Builder** window, click **VdcSP vs. Gas Flow**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Upper right**.

4 In the **VdcSP vs. Gas Flow** toolbar, click **Plot**.

From the results of Study 1, create a surface plot of the displacement.

Displacement, Without Gas Flow

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Displacement, Without Gas Flow` in the **Label** text field.

Surface 1

1 Right-click **Displacement, Without Gas Flow** and choose **Surface**.

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

3 In the **Expression** text field, type `solid.disp`.

4 From the **Unit** list, choose **µm**.

5 Select the **Description** checkbox.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Deformation 1

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

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

3 Select the **Scale factor** checkbox. In the associated text field, type `0`.

4 Locate the **Expression** section. Select the **Description** checkbox.

5 In the **Displacement, Without Gas Flow** toolbar, click **Plot**.

From the results of Study 2, create a surface plot of the displacement.

Displacement, With Gas Flow

1 In the **Results** toolbar, click **2D Plot Group**.
2 In the **Settings** window for **2D Plot Group**, type `Displacement, With Gas Flow` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2 - With Gas Flow/ Solution 2 (sol2)**.

Surface 1

1 Right-click **Displacement, With Gas Flow** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `solid.disp`.
4 From the **Unit** list, choose **µm**.
5 Select the **Description** checkbox.
6 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Deformation 1

1 Right-click **Surface 1** and choose **Deformation**.
2 In the **Settings** window for **Deformation**, locate the **Expression** section.
3 Select the **Description** checkbox.
4 Locate the **Scale** section.
5 Select the **Scale factor** checkbox. In the associated text field, type `0`.
6 In the **Displacement, With Gas Flow** toolbar, click **Plot**.
7 Click the **Zoom Extents** button in the **Graphics** toolbar.

From the results of Study 3, plot the wafer temperature versus time.

Wafer Temperature vs. Time

1 In the **Results** toolbar, click **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, type `Wafer Temperature vs. Time` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Wafer Temperature vs. Time/Solution 3 (sol3)**.

Point Graph 1

1 Right-click **Wafer Temperature vs. Time** 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 T.

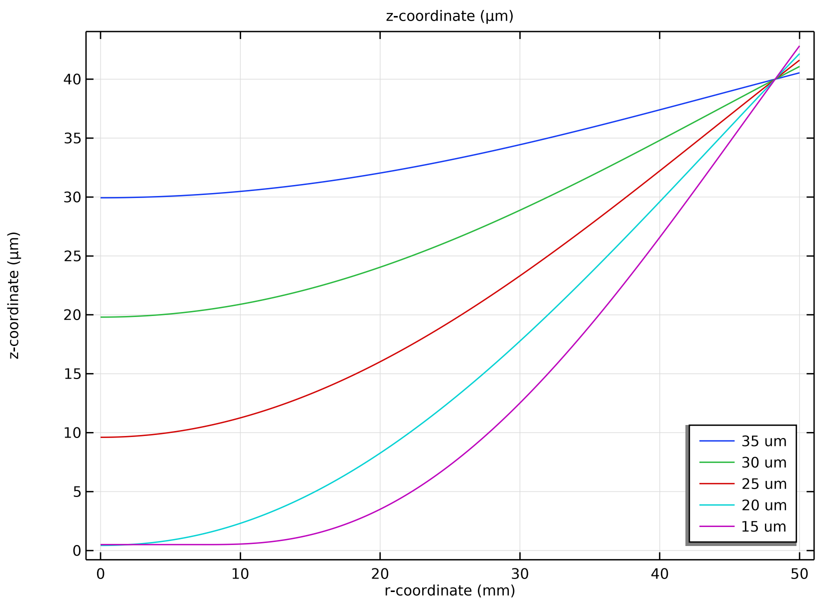
5 Select the **Description** checkbox.

6 Locate the **x-Axis Data** section. From the **Unit** list, choose **s**.

7 In the **Wafer Temperature vs. Time** toolbar, click **Plot**.

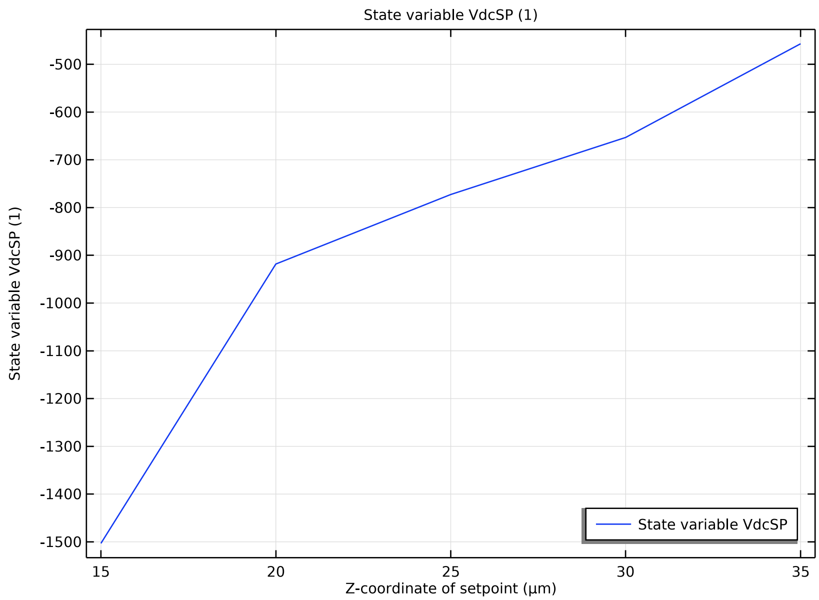
Line Graph 1

In the **Model Builder** window, under **Results** > **Wafer Profile, Without Gas Flow** click **Line Graph 1**.



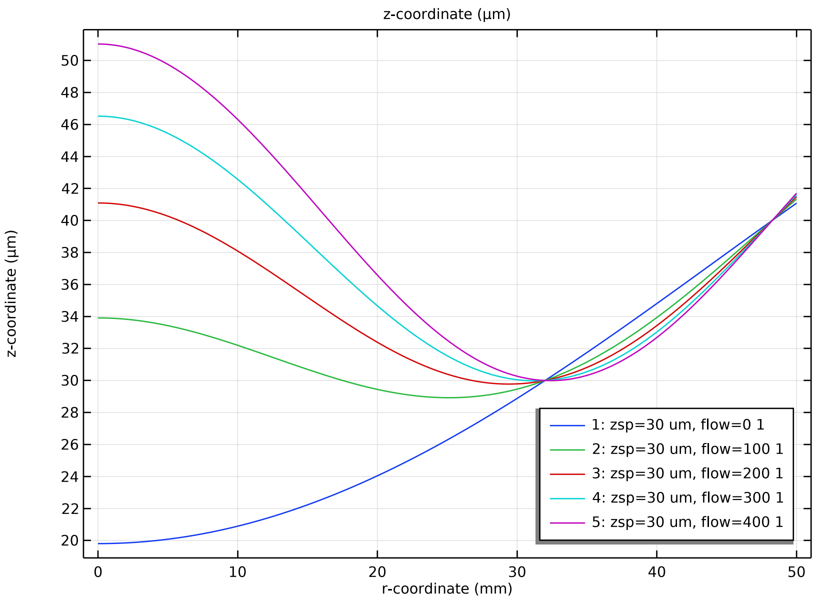
Global 1

In the **Model Builder** window, under **Results** > **VdcSP vs. zsp** click **Global 1**.



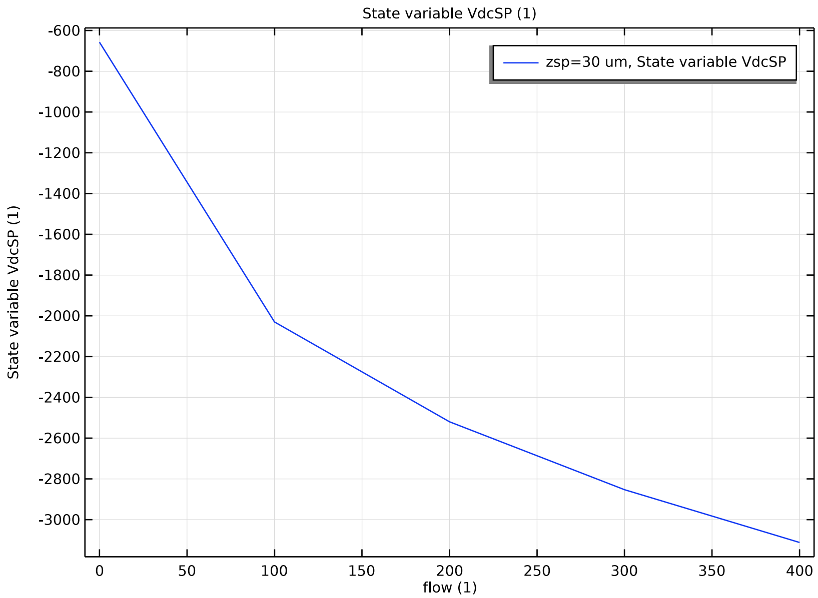
Line Graph 1

In the **Model Builder** window, under **Results > Wafer Profile, With Gas Flow** click **Line Graph 1**.



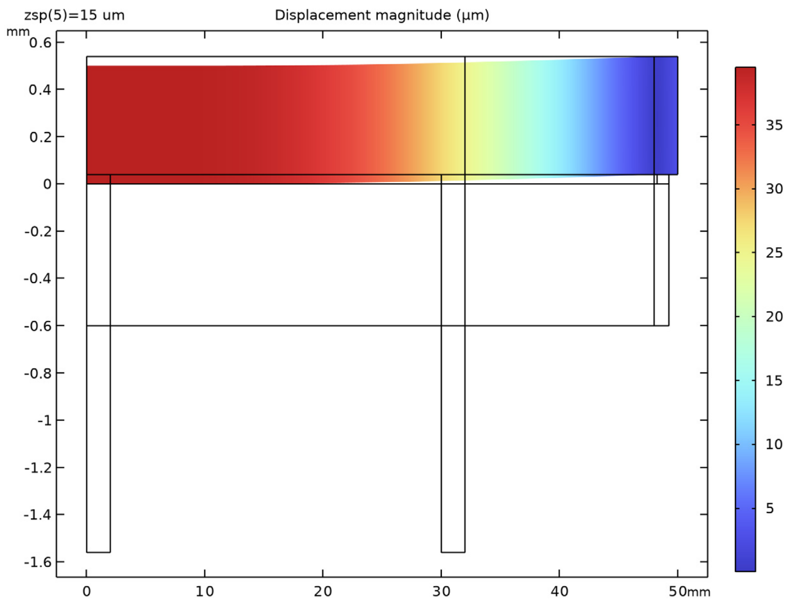
Global 1

In the **Model Builder** window, under **Results > VdcSP vs. Gas Flow** click **Global 1**.

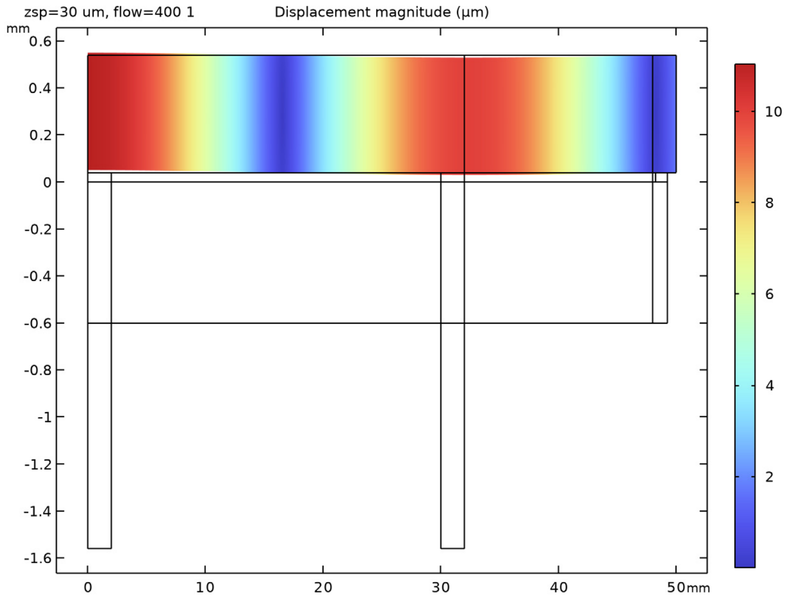


Surface 1

In the **Model Builder** window, under **Results** > **Displacement, Without Gas Flow** click **Surface 1**.



In the **Model Builder** window, under **Results** > **Displacement, With Gas Flow** click **Surface 1**.



Point Graph 1

In the **Model Builder** window, under **Results** > **Wafer Temperature vs. Time** click **Point Graph 1**.