



Model created in COMSOL Multiphysics 6.3

Thermal Expansion of a Rotary Vane Pump

Will Castro, Liv Melvin & Jos Moresi

ME526 Simulation of Physical Processes · Boston University · Fall 2025

Introduction

Rotary vane pumps are implemented in various environments, ranging from automobiles to drink dispensers. They can perform as high-, mid-, and low-range pressure pumps, depending on the application; vane pumps can even serve as vacuum pumps when pressures on the order of 10^{-6} bar are useful.⁽¹⁾

Consequently, analyses of these structures are becoming increasingly relevant as the reach of their applications and environments broadens.

In this example, the inner surface of a rotary vane pump's housing is subjected to a heat source (in the form of heated PVA glue). A 2D simplified model is used and is sufficient for estimation of expansion for our purposes. The structure is analyzed from thermal and structural points of view in order to determine the stress profile and extent of deformation.

The example utilizes the COMSOL-prescribed values for the density, heat capacity, and thermal conductivity of cast iron; the values for the thermal expansion coefficient, Poisson's ratio, and Young's modulus were estimated from publicly available data on gray iron.⁽²⁾ Further, general equations for thermal expansion of a pipe were used as a rough reference when estimating an acceptable range of output values.⁽³⁾

Model Definition

The simplified geometry of the pump is composed of a rectangular base attached to a ring which represents the pump's housing. The width of the ring is 0.55 inches, with an inner diameter of 3.6 inches and an outer diameter of 4.7 inches. The geometry of the structure is shown in [Figure 1](#).

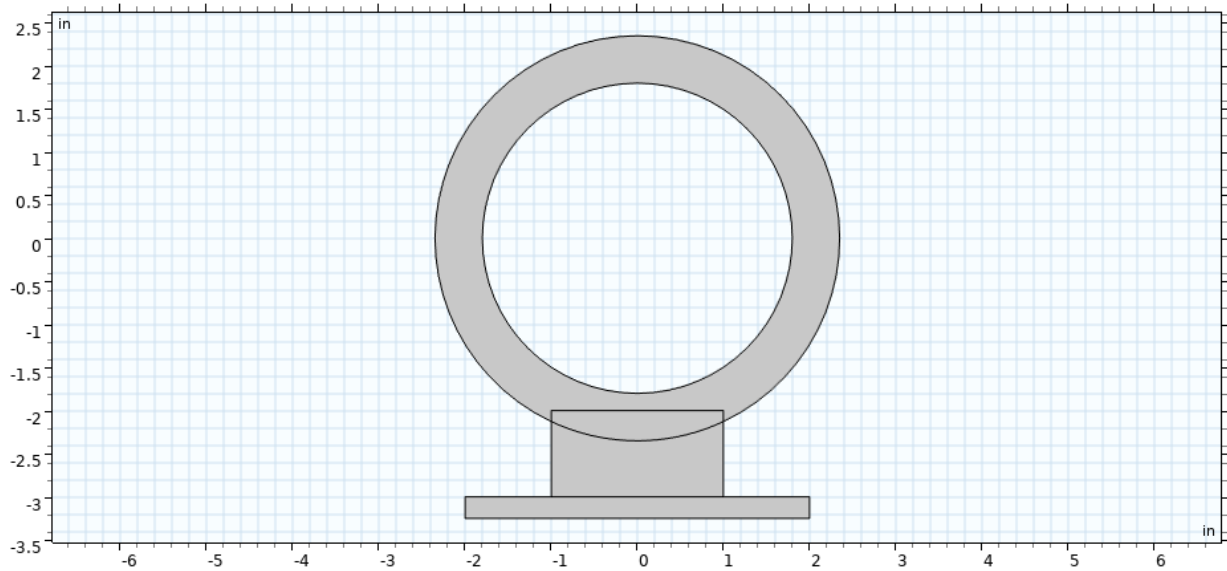


Figure 1: Simplified geometry of the rotary vane pump housing and base.

MATERIAL PROPERTIES

The entirety of the structure is made of the same material, assigned to the Cast Iron Material in COMSOL Multiphysics. The thermal expansion coefficient, Poisson's ratio, and Young's modulus properties of the material were estimated and input manually from publicly available data on gray iron. See the Introduction section for more information.

The material properties are given in [Table 1](#).

TABLE 1: MATERIAL PROPERTIES OF PUMP.

Material Properties	Value	Description
α	11E-6[1/K]	Coefficient of thermal expansion
ρ	7000[kg/m ³]	Density
C_p	420[J/(kg·K)]	Heat capacity at constant pressure
k	50[W/(m·K)]	Thermal conductivity
E	140[GPa]	Young's modulus

ν	0.25	Poisson's ratio
-------	------	-----------------

BOUNDARY CONDITIONS AND LOADS

The following boundary conditions and loads are applied to the model:

- Structural boundary conditions: The edge at $Y = -3.25$ in is fixed. This represents the base of the housing. All other applicable internal/external edges identified as free boundaries.
- Thermal boundary conditions: All external applicable edges are identified as a thermal insulator. This eliminates heat transfer with the surrounding environment. Our fluid operating temperature (heated PVA glue at 140°F) is applied to the boundaries that make up the inner diameter of the housing.

Results and Discussion

The temperature profile in the structure is uniform and shown in [Figure 2](#).

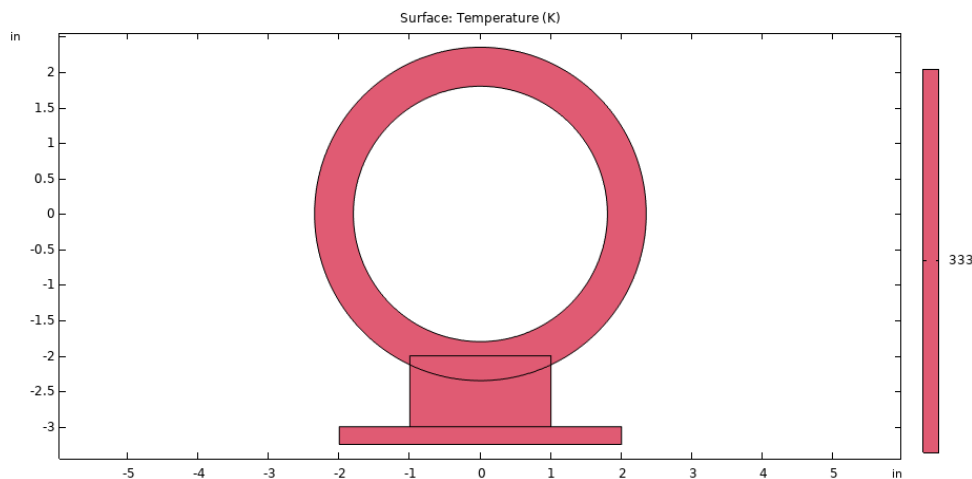


Figure 2: Temperature profile of the pump with a uniform temperature of 333K (140°F).

The effects of the thermal influence are evident in the von Mises stress gradient as shown in [Figure 3](#) and the deformation pattern as shown in [Figure 4](#). Note that the areas experiencing the most stress are located at the corners of the base and are not relevant to analysis of the thermal expansion of the pump's housing. Further, the stress concentrated at the inner, opposite surfaces of the housing

are influenced by their spatial arrangement relative to the fixed base constraint. Similarly, the structure's displacement is greatest in magnitude at the furthest point from its constrained base.

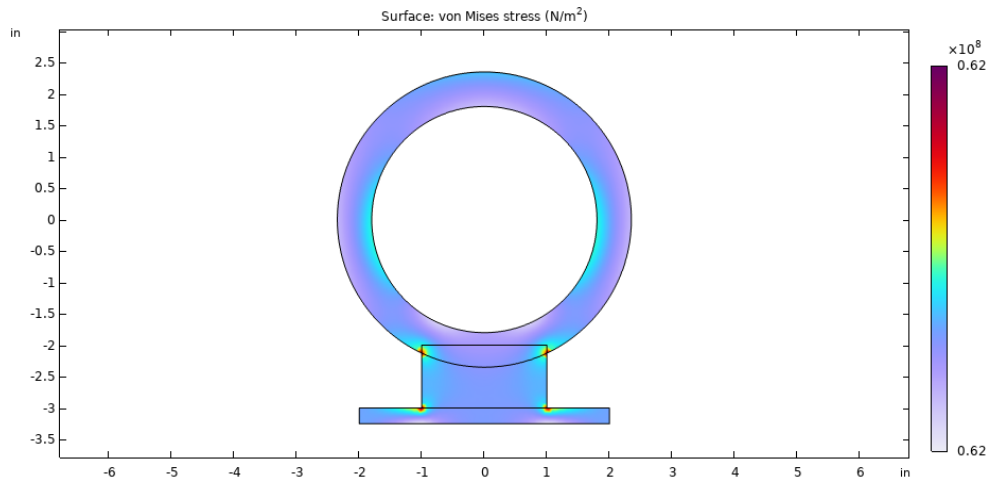


Figure 3: von Mises stress of entire structure with amplified color scale.

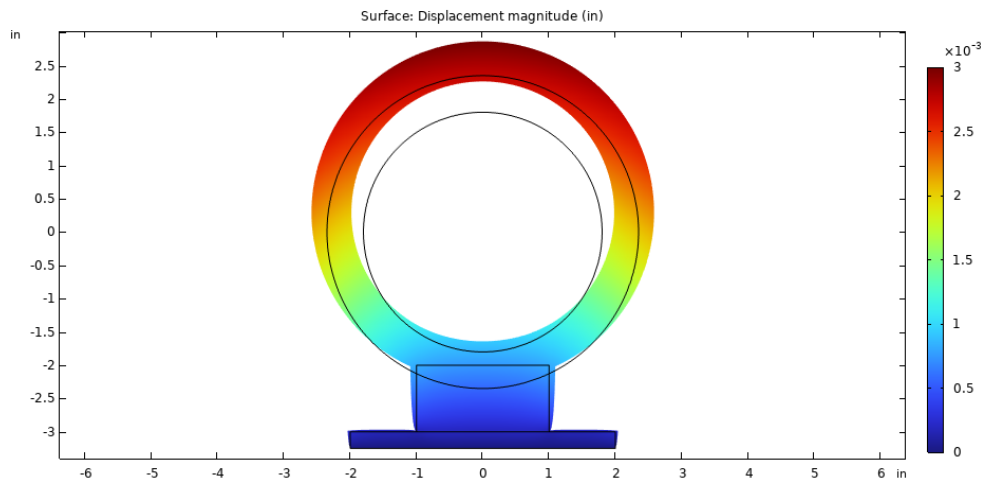


Figure 4: Displacement magnitude across pump resulting from thermal expansion.

The analysis of deformation can be compared to an axisymmetric model of the ring geometry alone and simplified to be approximated with a linear thermal expansion equation. As shown in [Figure 5](#), this process yields similar displacement values and can provide a useful reference for users while defining a range of reasonable results from the more complex model.

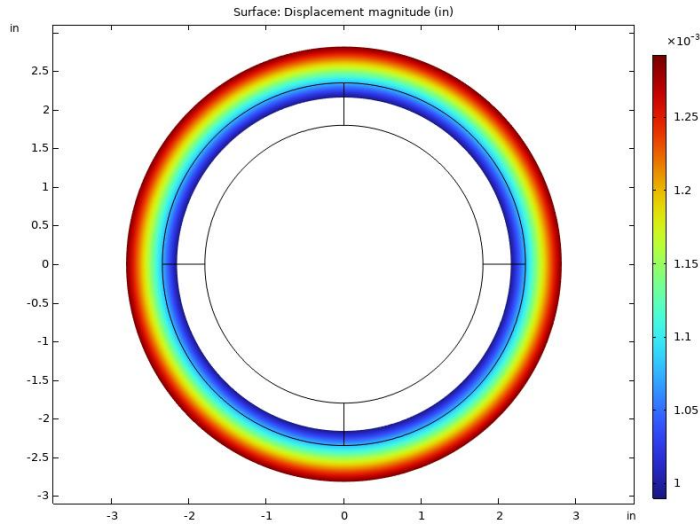


Figure 5: Radial displacement across housing respective to central axis without inclusion of pump base.

Notes About the COMSOL Implementation

In COMSOL Multiphysics, a structural analysis of a solid can be carried out using the Structural Mechanics (solid) Interface in the Structural Mechanics Module. The thermal analysis of the object can be carried out using the Heat Transfer in Solids Interface available in the Heat Transfer Module.

Modeling Instructions

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

NEW

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

MODEL WIZARD

- 1 In the **Model Wizard** window, click  **2D**.
- 2 In the **Select Physics** tree, select **Structural Mechanics** > **Heat Transfer** > **Heat Transfer in Solids**.
- 3 Click **Add**.
- 4 In the **Select Physics** tree, select **Structural Mechanics** > **Structural Mechanics (solids)**.
- 5 Click **Add**.
- 6 Click  **Study**.
- 7 In the **Select Study** tree, select **General Studies** > **Stationary**.
- 8 Click  **Done**.

GLOBAL DEFINITIONS

Parameters: [Thermal](#)

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 In the table, enter the following settings:

Name	Expression	Value	Description
T_fluid	333.15 [K]	333.15 K	Temperature of fluid
T_initial	294.261 [K]	294.261 K	Initial temperature

GEOMETRY 1

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

- 2 In the **Settings** window for **Geometry**, locate the **Units** section.
- 3 From the **Length unit** list, choose **in**.

Circle 1 (c1)

- 1 In the **Geometry** toolbar, click **Circle**.
- 2 In the **Settings** window for **Circle 1**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type $4.7/2$.
- 4 Click **Build All Objects**.

Circle 2 (c2)

- 1 In the **Geometry** toolbar, click **Circle**.
- 2 In the **Settings** window for **Circle 2**, locate the **Size and Shape** section.
- 3 In the **Radius** text field, type $3.6/2$.
- 4 Click **Build All Objects**.

Rectangle 1 (r1)

- 1 In the **Geometry** toolbar, click **Rectangle**.
- 2 In the **Settings** window for **Rectangle 1**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 2.
- 4 In the **Height** text field, type 1.
- 5 Locate the **Position** section. In the **y** text field, type -2.5.
- 6 Click **Build All Objects**.

Rectangle 2 (r2)

- 1 In the **Geometry** toolbar, click **Rectangle**.
- 2 In the **Settings** window for **Rectangle 2**, locate the **Size and Shape** section.
- 3 In the **Width** text field, type 4.
- 4 In the **Height** text field, type 0.25.
- 5 Locate the **Position** section. In the **y** text field, type -3.125.
- 6 Click **Build All Objects**.

Difference 1 (dif1)

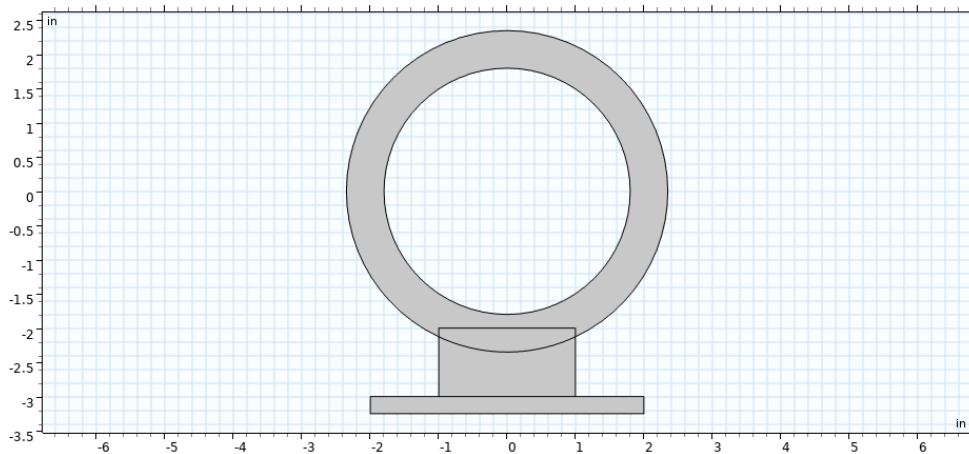
- 1 In the **Geometry** toolbar, click **Booleans and Partitions**. From the dropdown menu, select **Difference**.
- 2 In the **Graphics** window, select object **c1** only.
- 3 In the **Settings** window, activate the **Objects to subtract** selection.
- 4 In the **Graphics** window, select object **c2** only.
- 5 Click **Build All Objects**.

Form Union (fin)

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

2 In the **Settings** window, click **Build All**.

The geometry should look like the figure below.



Material: Cast Iron

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

2 In the **Add Material** window, choose **Built-in > Cast Iron**.

3 Click **Add to Component**.

SOLID MECHANICS

Fixed Constraint

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Fixed Constraint**.

2 In the **Graphics** window, select boundary 2 only.

HEAT TRANSFER

Initial Values

1 In the **Model Builder** window, under **Heat Transfer in Solids (ht)** click **Initial Values 1**.

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

3 In the **Temperature** text field, type T_{initial} .

4 Locate **Domain Selection**.

5 Under **Domain Selection** select **Activate Selection**.

Temperature

- 1 In the **Model Builder** window, under **Heat Transfer in Solids (ht)** click **Temperature 1**.
- 2 In the **Settings** window for **Temperature**, locate the **Temperature** section.
- 3 In the **Temperature** text field, type T_fluid.
- 4 Locate **Boundary Selection**.
- 5 Under **Boundary Selection** select **Activate Selection**.
- 6 Select boundaries 14, 15, 18, and 19 only.

MESH 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
- 2 In the **Settings** window for **Mesh 1**, locate the **Physics-Controlled Mesh** section.
- 3 From the **Element size** list, choose **Extra fine**.

STUDY 1

- 1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, click **Compute**.

RESULTS

Stress (solid)

- 1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.
- 2 In the **Stress (solid)** toolbar, click  **Plot**.

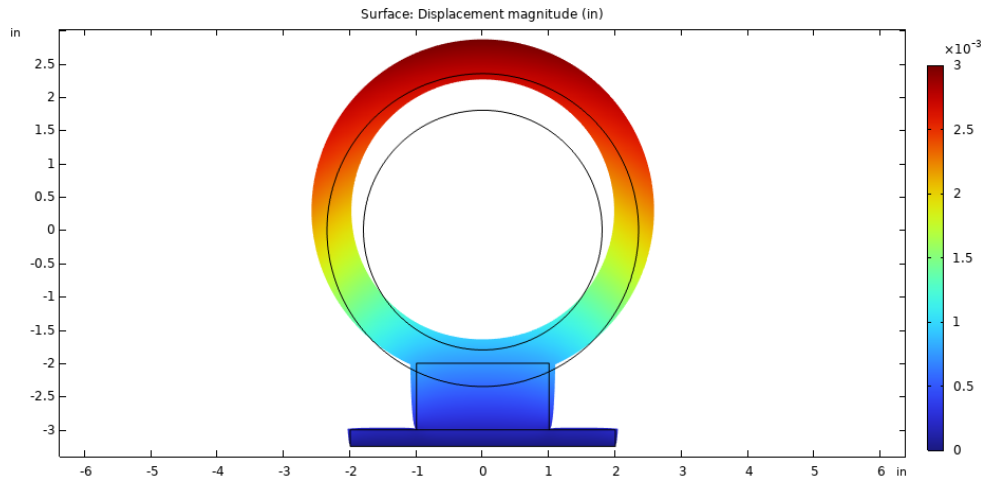
Temperature (ht)

- 1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface 1**.
- 2 In the **Temperature (ht)** toolbar, click  **Plot**.

2D Plot Group 3

- 1 In the **Model Builder** window, right click the **Results** node, then click **2D Plot Group**.
- 2 In the **2D Plot Group 3** toolbar, click **Surface**.
- 3 In the **2D Plot Group 3** toolbar, under the **Attributes** section, click **Deformation**.
- 4 Click  **Plot**.

The plot should look like the figure below.



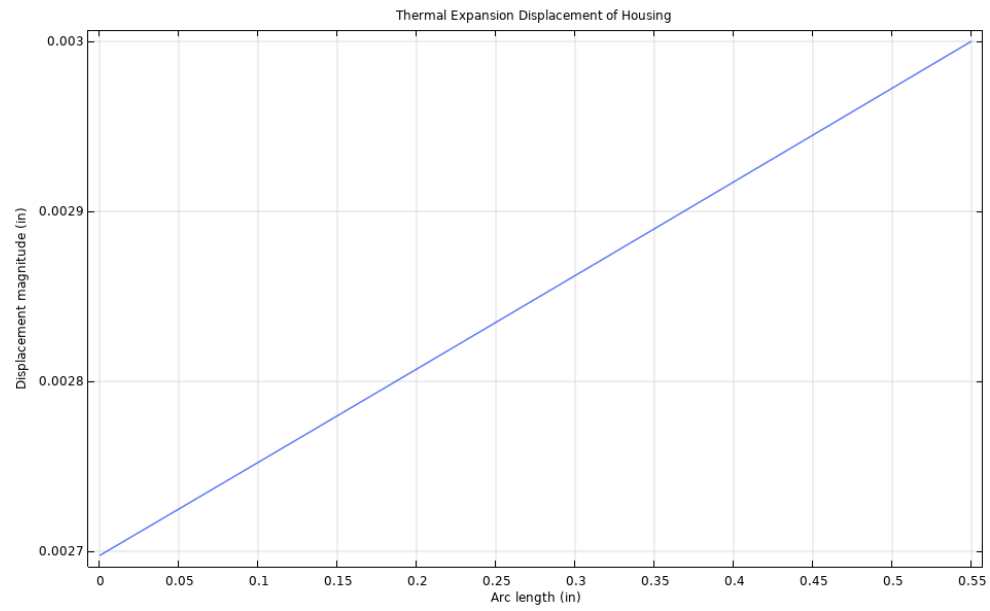
Cut Line 2D I

- 1 In the **Model Builder** window, under **Results**, right click the **Datasets** node, then click **Cut Line 2D**.
- 2 In the **Settings** window for **Cut Line 2D I**, locate the **Line Data** section.
- 3 In the **Point 1, Y** text field, type 1.8.
- 4 In the **Point 2, X** text field, type 0. In the **Point 2, Y** text field, type 2.35.
- 5 Click  **Plot**.

ID Plot Group 4

- 1 In the **Model Builder** window, right click the **Results** node, then click **ID Plot Group**.
- 2 In the **ID Plot Group 4** toolbar, click **Line Graph**.
- 3 In the **Settings** window for **Line Graph I**, locate the **Data** section.
- 4 From the **Dataset** dropdown menu, select **Cut Line 2D I**.
- 5 In the **Settings** window for **Line Graph I**, locate the **Title** section.
- 6 From the **Title type** dropdown menu, select **Manual**.
- 7 In the **Title** text field, type “Thermal Expansion Displacement of Housing”.
- 8 Click  **Plot**.

The plot should look like the figure below.



References

1. “Rotary vane pump.” Wikipedia.
https://en.wikipedia.org/wiki/Rotary_vane_pump
2. Penticton Foundry. “Gray Iron ASTM A48 Class 40.”
<https://www.pentictonfoundry.com/news/gray-iron-astm-a48-class-40/>
3. COMSOL. “Effects of Thermal Expansion.” COMSOL Multiphysics 6.1 Documentation, Structural Mechanics Module User’s Guide.
http://doc.comsol.com/6.1/doc/com.comsol.help.sme/sme_ug_pipe_mechanics.15.06.html