



VERSION 1

OCT 16, 2023

OPEN ACCESS



**DOI:**  
[dx.doi.org/10.17504/protocols.io.bp2l6x7dklqe/v1](https://dx.doi.org/10.17504/protocols.io.bp2l6x7dklqe/v1)

**Protocol Citation:** samuel.vorlet 2023. Simulating the modal analysis of hyperelastic membranes immersed in fluid using FE software ANSYS.  
**protocols.io**  
<https://dx.doi.org/10.17504/protocols.io.bp2l6x7dklqe/v1>

**License:** This is an open access protocol distributed under the terms of the [Creative Commons Attribution License](#), which permits unrestricted use, distribution, and reproduction in any medium, provided the original author and source are credited

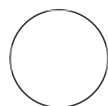
**Protocol status:** Working  
 We use this protocol and it's working

## 🌐 Simulating the modal analysis of hyperelastic membranes immersed in fluid using FE software ANSYS V.1

samuel.vorlet<sup>1,2</sup>

<sup>1</sup>Platform of Hydraulic Constructions (PL-LCH);

<sup>2</sup>Ecole Polytechnique Fédérale de Lausanne (EPFL)



samuel.vorlet

### ABSTRACT

This protocol provides step-by-step guidelines to perform the modal analysis of pre-strained hyperelastic rectangular membrane accounting for fluid-structure interactions using Finite Element software ANSYS. The nonlinear constitutive behavior of the material is modeled by the Mooney-Rivlin equation. The effect of water is considered by the added mass approach. The damping is included as Rayleigh damping.

**Created:** Oct 06, 2023

**Last Modified:** Oct 16, 2023

**PROTOCOL integer ID:**  
88896

## Hyperelastic material definition using the Mooney-Rivlin form...

- 1 Open **Engineering Data**.
- 2 Add a new material and define the material name. Verify the units.
- 3 Define the hyperelastic material properties.
  - 3.1 In the **Hyperelastic Experimental Data**, include the **Uniaxial Test Data**.
  - 3.2 In the **Uniaxial Test Data**, define the **Strain (mm<sup>-1</sup>)** and **Stress (Pa)** values from the uniaxial test data.
  - 3.3 In the **Hyperelastic** toolbox, choose the **Mooney-Rivlin** material model with the appropriate number of parameters (double-click on it). The number of parameters for the Mooney-Rivlin depends on the shape of the stress-strain curve and the number of inflexion points.
  - 3.4 In the **Curve Fitting** toolbox, choose the **Mooney-Rivlin X parameters** to add it to the material properties. Solve the curve fitting for the uniaxial test data (right-click on **Curve Fitting** -> **Solve Curve Fit**). Then, transfer the calculated values to the Mooney-Rivlin material constants (right-click on **Curve Fitting** -> **Copy Calculated Values to Property**). Verify that the material

constants are correctly inserted. Verify the Incompressibility Parameter ( $D1 = 0$  for incompressible materials).

- 4 Define the material density. In the **Physical Properties** toolbox, include the **Density** and define it.

When considering the added mass effect, the effective density should be defined. The added mass relies on the geometry and dimensions of the specimen and the density of the fluid  $\rho^f$ . For a rectangular specimen:

$$m_a = \rho^f \pi \left(\frac{c}{2}\right)^2 \left(\frac{s}{2}\right) K$$

where  $K$  is a constant and depends on the ratio  $c/s$  (sides of the specimen). The added density is obtained by dividing the added mass by the volume of the specimen:

$$\rho_a = \frac{m_a}{V_a} = \rho^f \pi \left(\frac{c}{8h}\right) K$$

Last, the effective density  $\rho_e$  is defined as:

$$\rho_e = \rho_s + \rho_a$$

where  $\rho_s$  is the structural density.

## Static analysis to define the initial conditions for the modal a...

- 5 Drag the **Static Structural** analysis from the toolbox over the **Engineering Data** module. Open the static structural analysis by double-clicking on the **Setup** component.
- 6 Draw or import the **Geometry**.
- 7 Define the analysis setup by opening the **Model** (starting ANSYS Mechanical).
- 7.1 Assign the material to the body. Select the **Geometry** in the outline, then select the **SYS\body**. Under **Details of SYS\body**, assign the **Material** by selecting the previously defined material.
- 8 Define the mesh properties by selecting the **Mesh** in the outline. Verify the **Physics Preference** (Mechanical) and the **Element Order** (Quadratic). In the Sizing, define the appropriate characteristics. The mesh can be refined by adding elements to the mesh characteristics (right-click on the **Mesh** in the outline -> **Insert** -> **Sizing/Refinement/Inflation/etc.**).

8.1 Define the element type for the meshing by selecting the **Geometry** in the outline, select the **SYS\body** and insert an APDL commande snippet (right-click -> Insert -> **Commands**). Use the following APDL code to define the desired mesh element type (for instance SOLID187):  
ET,matid,187  
(Ref:[https://www.mm.bme.hu/~gyebro/files/ans\\_help\\_v182/ans\\_elem/Hlp\\_E\\_SOLID187.html](https://www.mm.bme.hu/~gyebro/files/ans_help_v182/ans_elem/Hlp_E_SOLID187.html))

9 Define the analysis setup by selecting **Static Structural** in the outline.

9.1 Activate nonlinear analysis by turning ON the **Large Deflection** in the **Solver Controls**.

9.2 Insert gravity effects on the **Analysis Settings** in the outline (right-click -> Insert -> **Standard Earth Gravity**). Define the desired **Z Component** of the gravity ( $\text{m/s}^2$ ).

9.3 Define the boundary conditions by selecting the **Analysis Settings** in the outline (right-click -> Insert -> **Fixed Supprt/Displacement/etc.**). Assign the boundary to the geometry part (**Geometry Selection**) and define the boundary type (**Fixed Support** for instance).

9.4 Define the external load (hydrostatic pressure) by selecting the **Analysis Settings** in the outline (right-click -> Insert -> **Hydrostatic Pressure**). Select the face where to apply the hydrostatic pressure (**Scope -> Geometry**), define the fluid density (**Definition -> Fluid Density**) and the magnitude of the hydrostatic pressure (**Free Surface Location -> Z Coordinate**).

9.5 Define the pre-strain by selecting the **Geometry** in the outline, select the **SYS\body** and insert an APDL command snippet using the INISTATE command (right-click -> Insert -> **Commands**). The following APDL code must be used:  
cmsel,s,body,elem !Geometry selection (here body, to be adapted)  
INISTATE,SET,DTYP,STRE !Set stress value, in Pa  
INISTATE,DEFINE,,,,,500000,0,0,,, !Define the stress value (here, 0.5 MPa)  
ALLSEL

10 Run the static analysis by selecting the **Static Structural** in the Outline (right-click -> **Solve**).

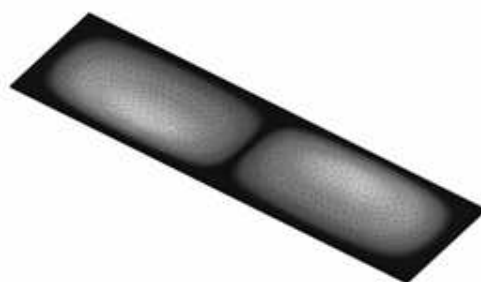
- 11 Visualize the results by selecting **Solution** in the outline (right-click -> Insert -> **Deformation/Strain/Stress/etc.**). After adding the results type, right-click on **Solution** in the Outline and select **Evaluate All Results**.

## Modal analysis of a pre-strained hyperelastic material

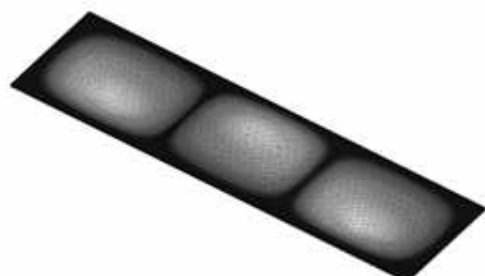
- 12 Drag the **Modal** analysis from the toolbox over the **Solution** of the **Static Structural** module. Open the modal analysis by double-clicking on the **Setup** component.
- 13 Define the number of modes to calculate by selecting the **Analysis Settings** in the **Modal** in the Outline. In **Details of Analysis Settings**, define the **Max Modes to Find**.
- 14 When considering the damping effect, the damping must be inserted by selecting **Analysis Settings** in the Outline and **Solver Controls** -> **Damped** -> Yes. Then, the damping parameters must be given under **Damping Controls**.  
The damping is defined as Rayleigh damping (linear combination of mass and stiffness):  
$$[c] = \alpha_m [m] + \beta_k [k]$$
  
The coefficients are frequency-dependent and are given by:  
$$\alpha_m = \frac{2\omega_1\omega_2(\xi_1\omega_2 - \xi_2\omega_1)}{\omega_2^2 - \omega_1^2}$$
  
$$\beta_k = \frac{2(\xi_2\omega_2 - \xi_1\omega_1)}{\omega_2^2 - \omega_1^2}$$
  
where  $\omega_n$  is the  $n$ -th mode natural circular frequency in vacuum and  $\xi_n$  is the corresponding damping ratio. The  $n$ -th natural frequencies are determined using the FE numerical model in vacuum.
- 15 Run the modal analysis by selecting the **Modal** in the Outline (right-click -> **Solve**). The natural frequencies can be found by clicking on the **Solution Information** of the **Solution**.
- 16 Visualize the results (mode shapes) by selecting **Solution** in the outline (right-click -> Insert -> **Total Deformation**) and choose the mode number by selecting **Total Deformation** and **Mode**. After adding the results type, right-click on **Solution** in the Outline and select **Evaluate All Results**.



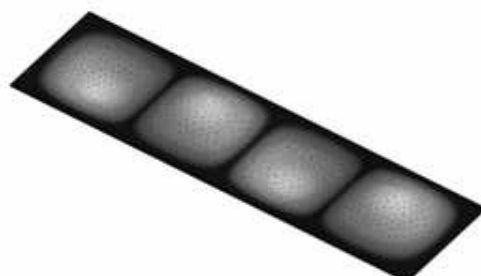
$$\Gamma_1 = 13.08 \text{ [Hz]}$$



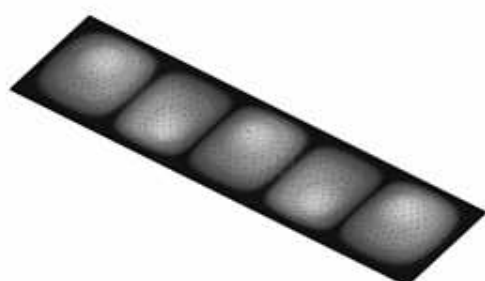
$$\Gamma_2 = 13.56 \text{ [Hz]}$$



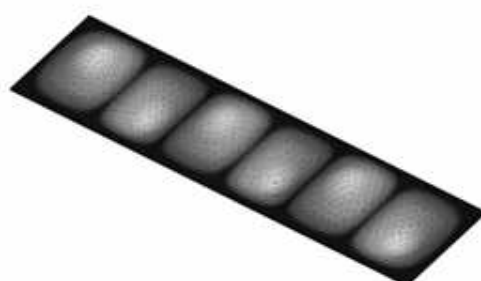
$$\Gamma_3 = 14.56 \text{ [Hz]}$$



$$\Gamma_4 = 16.28 \text{ [Hz]}$$



$$\Gamma_5 = 18.80 \text{ [Hz]}$$



$$\Gamma_6 = 22.10 \text{ [Hz]}$$