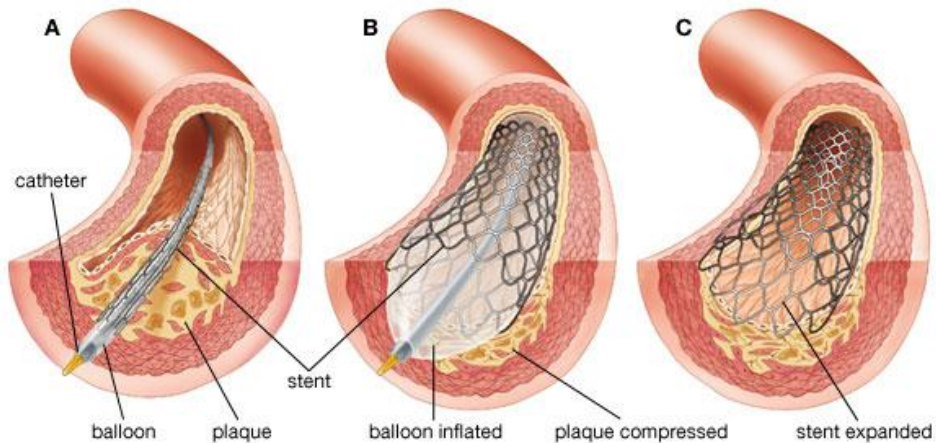
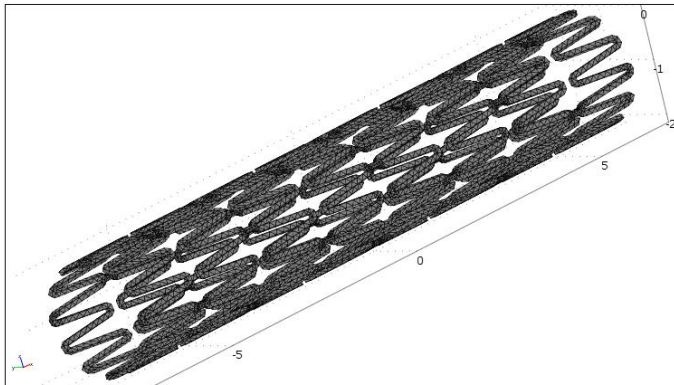


## MEDICAL STENT – STRUCTURE MECHANICS ANALYSIS

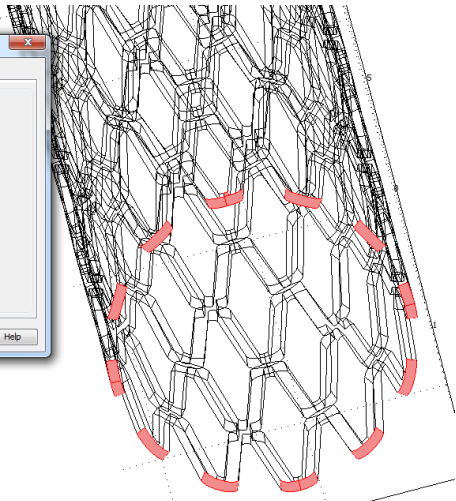
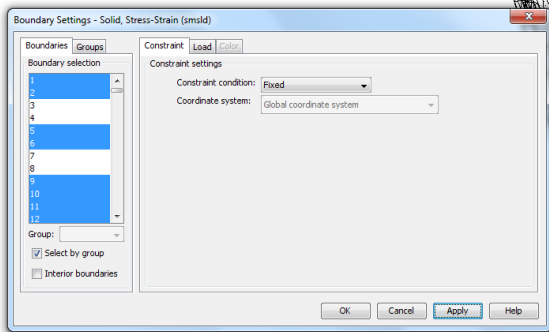


## GEOMETRY MODEL & MESH





## BOUNDARY CONDITIONS – TYPE 1- FIXED



## BOUNDARY CONDITIONS – TYPE 2 - LOAD

The screenshot displays the COMSOL Multiphysics interface with the **Boundary Settings** dialog box open for a **Load** boundary condition. The dialog is titled "Boundary Settings - Solid, Stress-Strain (solid)".

**Boundary selection:** A list of boundaries is shown, with boundaries 7 and 8 selected.

**Load settings:**

- Type of load: Distributed load
- Coordinate system: Tangent and normal coord. sys. ( $t_n, n_r$ )

**Quantity:** A table lists the quantities and their corresponding values and units:

Quantity	Value/Expression	Unit	Description
$F_t$	0	$\text{N/m}^2$	Face load (force/area) $t_t$ -dir.
$F_n$	0	$\text{N/m}^2$	Face load (force/area) $n$ -dir.
$F_s$	0	$\text{N/m}^2$	Face load (force/area) $s$ -dir.

**Buttons:** OK, Cancel, Apply, Help

The background shows a 3D model of a structure with a red mesh. The structure consists of a series of interconnected, curved elements forming a lattice-like pattern. The mesh is applied to the surface of these elements.

At the bottom of the window, the status bar indicates: "Saved COMSOL Model File: stent00.mph"

# NUMERICAL RESULTS

