Basic steps in modeling with Comsol Multiphysics

0. Start Comsol Multiphysics

PROBLEM DEFINING

1. Select space dimension
   - 0D, 1D,2D,3D or axisymmetric ?

2. Add physics

3. Select study type
   - Stationary, time dependent, …

4. Define geometry
   - Draw or import

5. Define parameters/variables (if such are needed)

6. Define materials

7. Define needed parameters for physics

8. Mesh geometry (Mesh / Build All)
   - Mesh type
   - Element size?

PROBLEM SOLVING

9. Solve problem (Study / Compute)
   - Linear, non-linear, stationary, parametric, time-dependent?

POSTPROSESSING

10. Analyze results (Postprocessing)
    - Plot parameters
    - Derived values