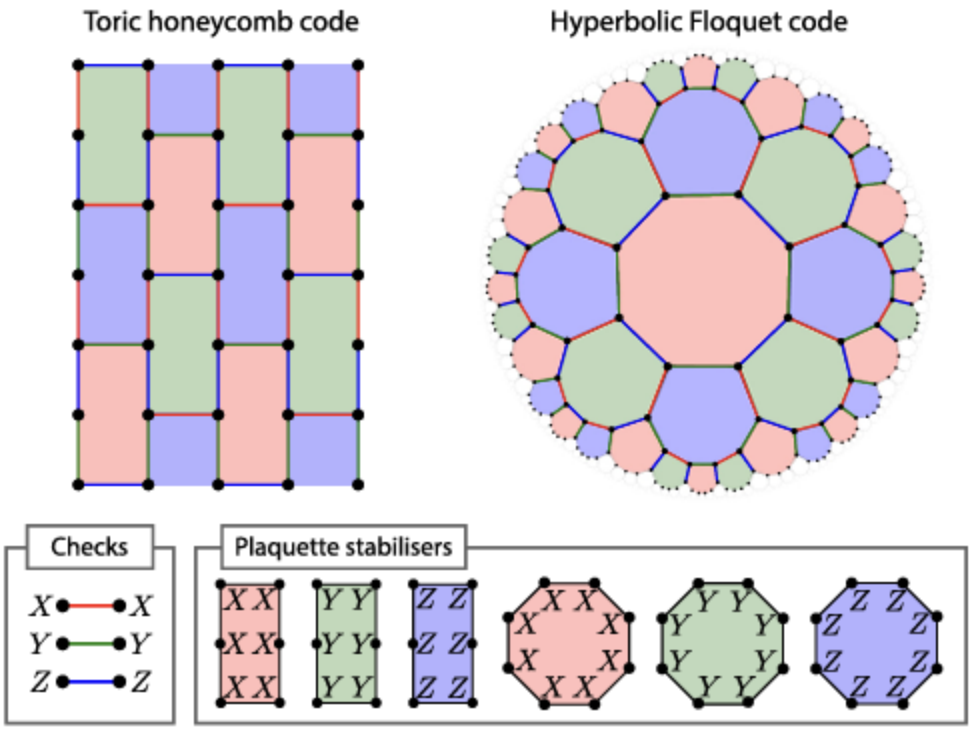

COMSOL Multiphysics to simulate a simple **electromagnetic field** and **quantum effects** in a material, inspired by the extreme environments of galactic objects like black holes or magnetars. The example will cover a simulation of a **magnetic field** in a quantum system, which can be extended to simulate the behavior of quantum circuits under extreme magnetic conditions.

Example: Simulating a Magnetic Field in a Quantum System Using COMSOL

1. Setting up a Model:

We will create a model that simulates the behavior of a **magnetic field** using the **AC/DC module** in COMSOL. The model will use a **3D geometry** to represent a magnetic field interacting with a quantum system or a material under extreme conditions like those near a magnetar.

2. Steps to Set Up the Simulation:

1. **Open COMSOL Multiphysics** and select the **3D Model**.
2. **Define Geometry:**
    - Create a **sphere** or **cube** to represent a material or space where the magnetic field will interact. This could represent a region around a neutron star or magnetar.
    - For this, go to **Geometry** > **Sphere** and set the radius to represent the scale of the region under study.
3. **Add Material Properties:**

    - Define the material properties of the region you're simulating. For a magnetic material, you might select a **ferromagnetic material** or a **superconductor**.
    - Go to **Materials** > **Add Material** > **Custom** and input properties like **magnetic permeability** or **susceptibility** that correspond to extreme magnetic environments.
4. **Create Magnetic Field:**
    - In the **Physics** interface, add a **Magnetic Fields (mf)** physics node. This allows you to define a **magnetic field** and simulate its effects on materials.
    - In **Magnetic Fields (mf)**, you can apply a **magnetic field** source, such as a uniform magnetic field or a dipole field (representing a magnet's magnetic field).
5. **Define Boundary Conditions:**
    - Set boundary conditions to simulate the interaction of the magnetic field with the material. For instance, a **magnetic insulation** or **magnetic permeability boundary** might be used to represent a material that is either perfectly insulating or highly permeable.
6. **Mesh the Geometry:**
    - Choose an appropriate mesh for the simulation. Since you're simulating a high-energy or high-magnetic field, you may want to use a finer mesh near boundaries or regions of interest.
7. **Simulation Setup:**
    - Choose **Stationary** or **Time-dependent** solvers depending on whether you are studying a steady-state or dynamic field.
8. **Run the Simulation:**
    - Solve the model to compute the distribution of the magnetic field in your material. You can visualize the magnetic field lines or scalar values of the field strength.

3. Visualizing Results:

After running the simulation, you can visualize the results using various **plot types** in COMSOL, such as:

- **Magnetic Field Lines**: To see how the magnetic field propagates through the material.
- **Magnetic Flux Density (B-field)**: To show the magnitude and direction of the magnetic field at various points in the material.
- **3D Surface Plot**: To view the magnetic field distribution in three dimensions.

Example Code for Simulating a Magnetic Field in COMSOL (Script Example)

While COMSOL typically uses a GUI interface for setting up models, you can also automate simulations using **COMSOL's Java API** or **MATLAB LiveLink**. Below is a simple example script using **MATLAB** to set up a **magnetic field** simulation in COMSOL.