Designing a **step-down converter (buck converter)** using **LTspice** to derive **24V, 12V, and 5V** from a **36V battery** for your bot involves several steps. Below is a detailed **step-by-step tutorial**, including **component selection** and **LTspice simulation setup**.

**Step 1: Understanding the Design**

We will design **three buck converters** to step down **36V** to **24V, 12V, and 5V**, ensuring:

* **High efficiency** (using switching regulators instead of linear regulators).
* **Stable voltage output** (using feedback control and capacitors).
* **Protection features** (MOSFET for power control).

**Step 2: Selecting Components**

For each voltage rail, we need a **buck converter topology**, consisting of:

1. **MOSFET Switch** (Controlled by a PWM signal)
2. **Inductor (L)**
3. **Diode (D)**
4. **Capacitors (C)**
5. **Feedback control (to regulate voltage)**

We will use **LTSpice models** for realistic simulation.

**Component Selection**

| **Component** | **Specification** |
| --- | --- |
| **Battery** | 36V DC source |
| **Switching MOSFET** | IRF540N (N-channel, low Rds(on)) |
| **Diode** | Schottky Diode 1N5819 (for fast switching) |
| **Inductors** | Calculated per voltage rail |
| **Capacitors** | 100µF for filtering |
| **Regulators** | LM2596 (for 5V and 12V) & LM338 (for 24V) |
| **Control PWM** | Generated in LTspice for MOSFET |

**Step 3: Designing the Buck Converters**

**Formula for Inductor & Capacitor Selection**

The standard buck converter equation is:

Vout=Vin×(D1)V\_{out} = V\_{in} \times \left(\frac{D}{1}\right)Vout​=Vin​×(1D​)

where:

* DDD = Duty cycle of PWM.
* Vin=36VV\_{in} = 36VVin​=36V.
* Vout=24V,12V,5VV\_{out} = 24V, 12V, 5VVout​=24V,12V,5V.

L=(Vin−Vout)×Dfsw×IoutL = \frac{(V\_{in} - V\_{out}) \times D}{f\_{sw} \times I\_{out}}L=fsw​×Iout​(Vin​−Vout​)×D​

where:

* fswf\_{sw}fsw​ = Switching frequency (typically 100kHz–200kHz).
* IoutI\_{out}Iout​ = Output current.

Using this, we get:

| **Voltage** | **Inductor (L)** | **Capacitor (C)** |
| --- | --- | --- |
| **24V** | 150µH | 100µF |
| **12V** | 220µH | 220µF |
| **5V** | 330µH | 470µF |

**Step 4: Designing the Circuit in LTspice**

1. **Add a 36V Battery Source**
   * Use voltage component and set **DC = 36V**.
2. **Add a Power MOSFET for Switching**
   * Use IRF540N as the **main switch** between battery and output.
   * Gate controlled by a **PWM source**.
3. **Add an Inductor for Each Output Voltage**
   * Use **150µH, 220µH, and 330µH** inductors.
4. **Add a Schottky Diode**
   * Place **1N5819** in **parallel** to inductor (cathode to output).
5. **Add Output Capacitors**
   * Use **100µF, 220µF, and 470µF** electrolytic capacitors.
6. **Add Load Resistors**
   * Set appropriate values for **24V, 12V, and 5V** outputs.
7. **Implement PWM Control**
   * Generate **PWM** (duty cycle adjusted for each output).
   * Connect to the **MOSFET gate**.
8. **Add a Mechanical Stop Switch**
   * Add a **MOSFET IRF540N** **between the battery and circuit**.
   * Control the **gate with a switch** (on/off).

**Step 5: Simulation in LTspice**

1. **Run a transient analysis** (.tran 50ms).
2. **Observe output voltages** at 24V, 12V, and 5V nodes.
3. **Check MOSFET switching** behavior.
4. **Modify PWM duty cycles if needed** for proper regulation.

**Step 6: Adding Voltage Regulators for Stability**

* Use **LM2596** (buck converter IC) for **12V and 5V**.
* Use **LM338** (adjustable regulator) for **24V**.

**Final Notes**

* **Ensure MOSFET can handle the load current**.
* **Use heatsinks** for regulators if needed.
* **Check current ratings of inductors and capacitors**.