Power Electronics with Wide Bandgap Devices Dr. Moumita das School of Computing and Electrical Engineering Indian Institute of Technology, Mandi

## Lecture-26 GETTING STARTED WITH LTSPICE - Part 01

## **GETTING STARTED WITH LTSPICE - PART 01**

**Refer Slide(0:00)** 



Hello, welcome to the course on power electronics with Wide Band Gap devices. Today I am going to discuss about the importing of wide band gap devices in LTSpice software. So let's see how we can import any device in LTSpice for the simulation use.

Refer Slide(0:47)



Okay, so this is the detailed procedure which I am going to show you the actual how it is can be done. So, first we need to go to the website of the Wolfspeed because the silicon carbide device which we are selecting that is from the Wolfspeed. So, let go to this particular website, www.wolfspeed.com/tools and support / power / LTSpice and plecs models. So, you can just directly go to the website and after going to the website, you can find tools and support, right? And then you can find power under tools and support and then under power you can actually go to LTSpice and Plecs models. This is the place where we are going directly at this moment.

## Refer Slide(1:37)



So once you go to this LTSpice and Plecs models, you will be able to see this particular page in the website, in the Wolfspeed website. So, now in this you can see there are different models are given, Plecs models are given, Spice models are given. Since we are using LTSpice, we are going to consider the Spice model. So, you need to open the Spice models.

## **Refer Slide(2:10)**

https://www.wolfspeed.com	n/tools-and-support/power/Itspice-and	-plecs-models/	Bi AN 🔿 Di	1 G () %
by downloading Lispice and PLECS models non-wonspeed, you agree to our <u>Terms or ose</u> and <u>Privacy Policy</u> . Downloads				
Product Type	Search For Documen	ts	Wolfspeed_SPICE_Models (1).zip	00 ×
View All Power			See more	
Materials				
Document Type	Document Typ	e Document Name		Last Updated
Models (6)	User Guide	PRD-07913: Wolfspeed Power Module SPICE Models Us	ser Guide	12/2023
User Guide (1)	our ouroe			IL/LOLD
Videos (2)	Models	All SiC MOSFET PLECS Models		06/2024
Format	Models	All PLECS Device Models		06/2024
3D step (1)	Modals			06/2024
Altium (1)	Models	a All SPICE Models		06/2024
EAGLE (1)	Models	All SiC Schottky Diode PLECS Models		05/2024
KiCad (1)				
LTspice (4)	Models	All SiC Module PLECS Models		02/2024
OrCad (1) PLECS (5)	Models	CRD25DA12N-FMC PLECS Models		10/2023
CLEAR ALL FILTERS	Videos	Analyzing Transient Behavior Using a Double Pulse Test Simulation in LTspice		07/2024
Topics	Videos	How to Simulate Silicon Carbide Power Module Performance with LTSpice		11/2023
*				

It will take time that particular some to go to page. So, then when you are downloading the spice model, so you can see this kind of zip file will be downloaded. So, then you go to the zip file to see what are the models given. It is taking some time to open the zip file. So you can actually go to other websites also to see if the SPICE model is given with respect to that particular device. Okay? So, you can see this is the folder which downloaded. is

Refer Slide(3:04)



Now, you can open this particular folder. So, under this folder different spice models are given. So, you go inside this folder, you can see these are the different folders you will be able to see. Under this, you can either select modules, discrete components, diodes, bare die, any type of component which you can.