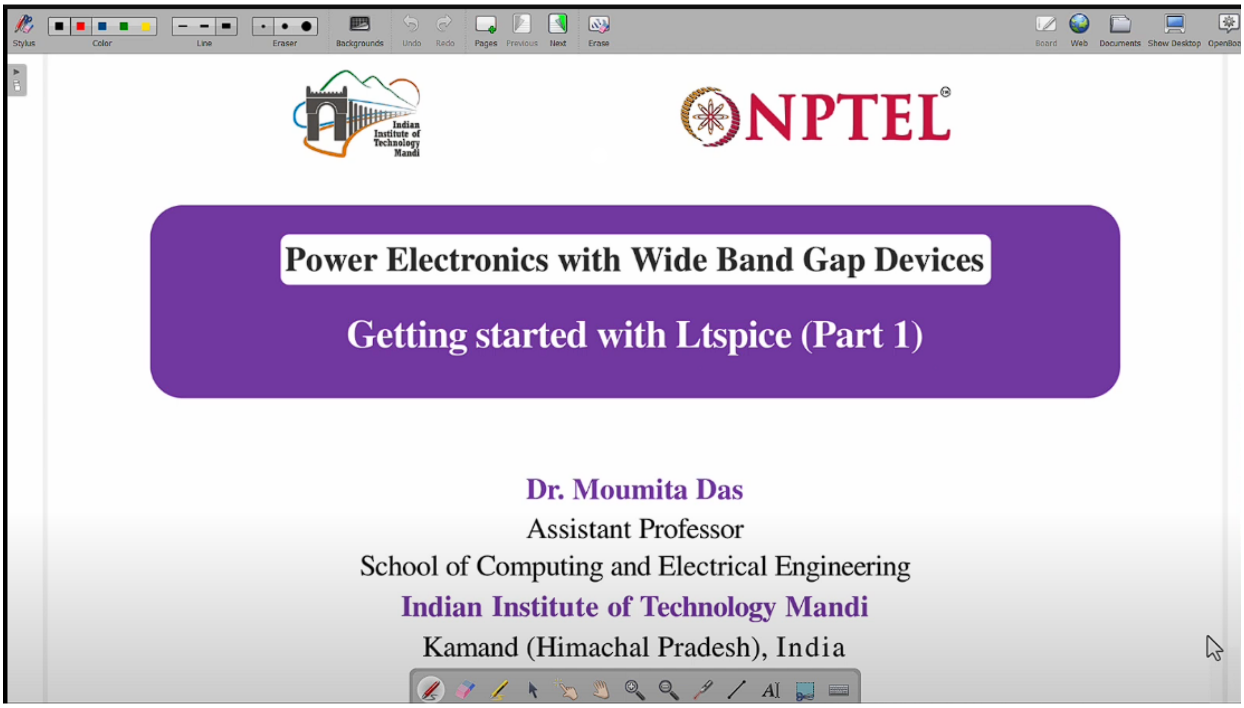
Power Electronics with Wide Bandgap Devices
Dr. Moumita das
School of Computing and Electrical Engineering
Indian Institute of Technology, Mandi

Lecture-26
GETTING STARTED WITH LTSPICE - Part 01

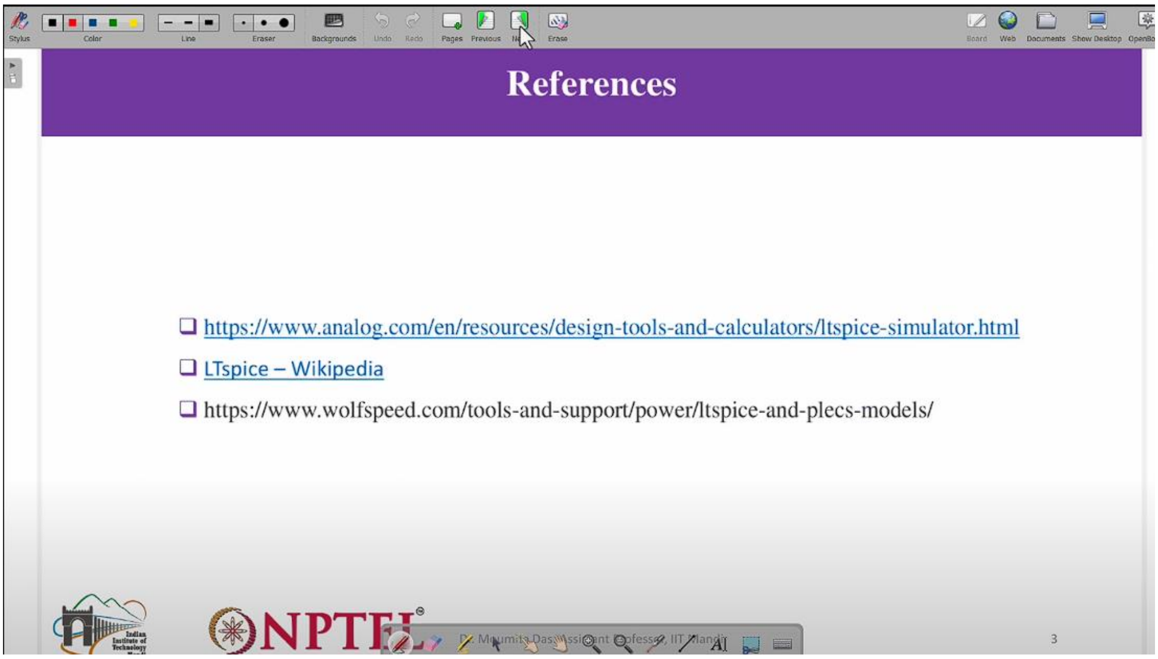
GETTING STARTED WITH LTSPICE - PART 01

Refer Slide(0:00)



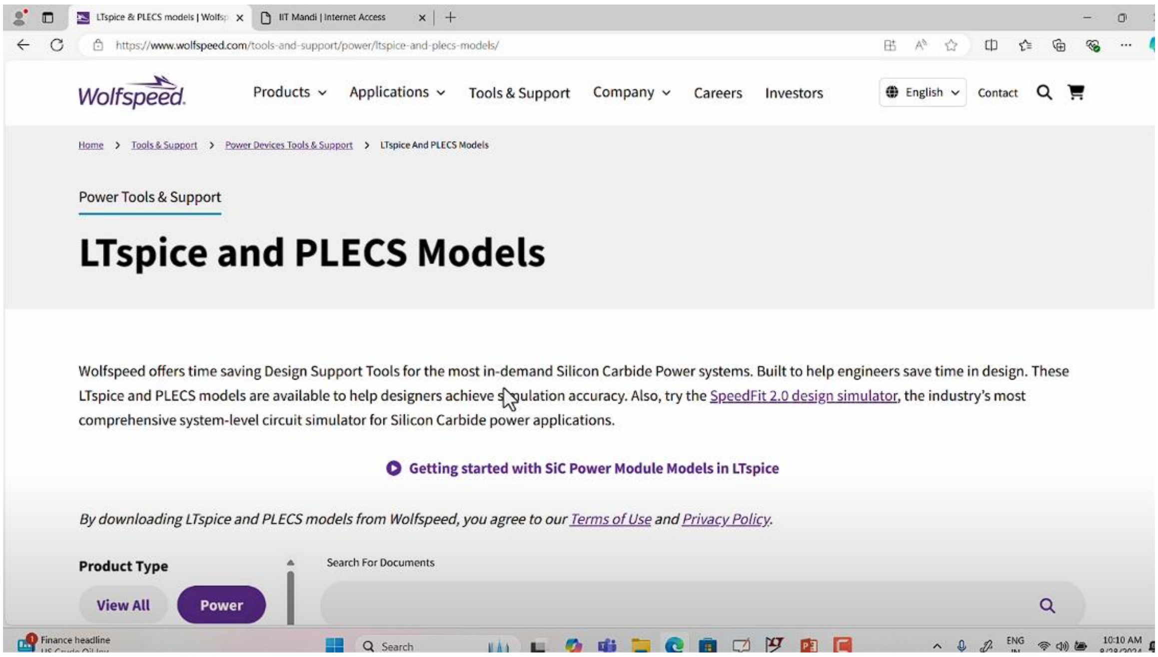
Hello, welcome to the course on power electronics with Wide Band Gap devices. Today I am going to discuss about the importing of wide band gap devices in LTSpice software. So let's see how we can import any device in LTSpice for the simulation use.

Refer Slide(0:47)



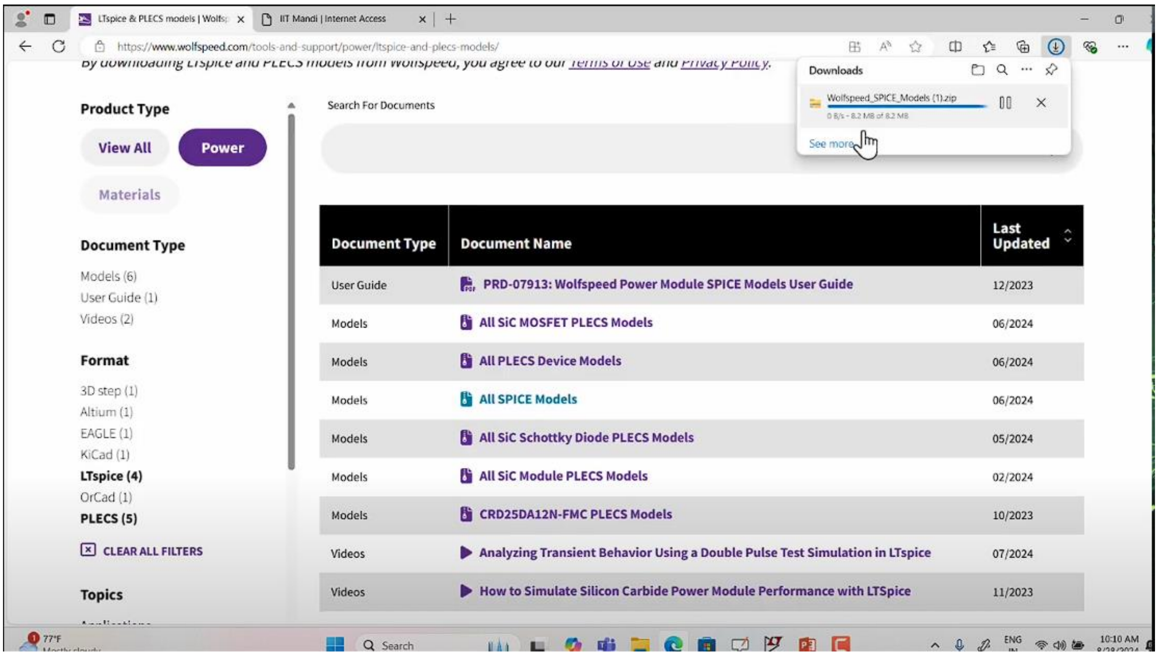
Okay, so this is the detailed procedure which I am going to show you the actual how it is can be done. So, first we need to go to the website of the Wolfspeed because the silicon carbide device which we are selecting that is from the Wolfspeed. So, let go to this particular website, www.wolfspeed.com/tools 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)



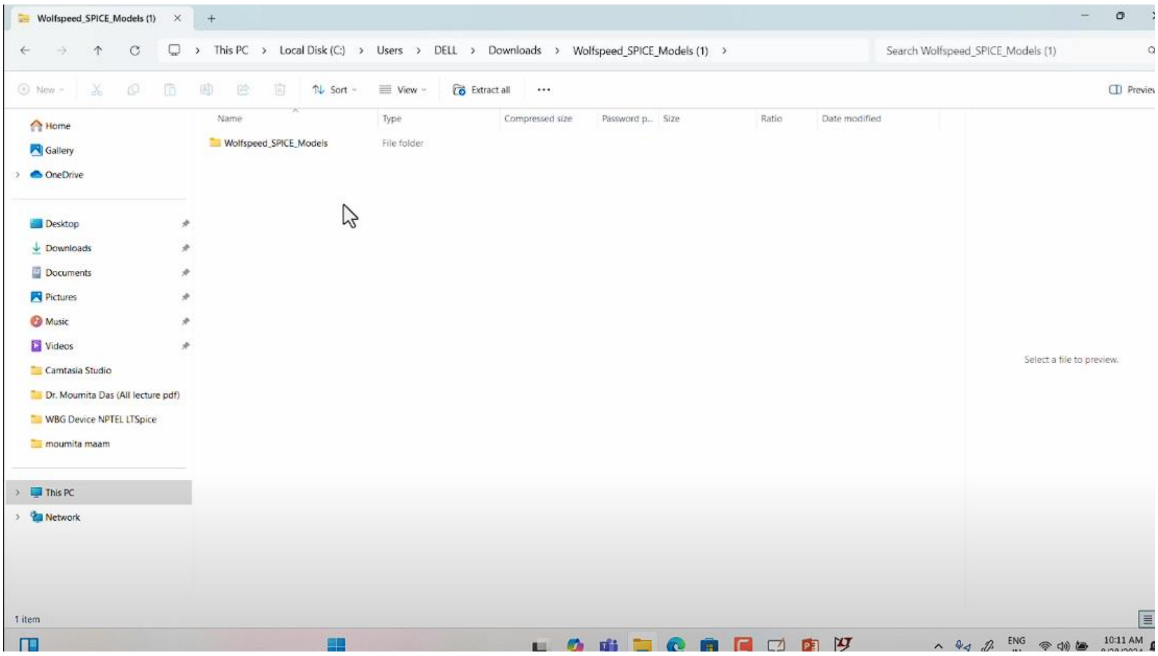
So once you go to this LTSpice and Plecs models, you will be able to see this particular page in the website, in the Wolfspeed website. So, now in this you can see there are different models are given, Plecs models are given, Spice models are given. Since we are using LTSpice, we are going to consider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

Refer Slide(2:10)



It will take some time to go to that particular 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 is downloaded.

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.