Be smart. Prototype online.

Infineon Designer
Getting Started

www.infineon.com/ifxdesigner

Get support: mailto:support@infineon.com

2019-Jan-21
Agenda

1. Infineon Designer SPICE Simulation Overview
2. Infineon Designer New Features
3. Getting Started Examples
4. Support
Agenda

1. Infineon Designer SPICE Simulation Overview
2. Infineon Designer New Features
3. Getting Started Examples
4. Support
Infineon Designer - Online Digital Prototyping Engine (www.infineon.com/ifxdesigner)

Great user experience

› New! Full-featured circuit editor
› Multiple platforms (IE, Safari, Chrome, Firefox, etc.)
› No installation
› Unlimited licenses
› Fast simulation due powerful server configuration

Features

› Accurate transient and system efficiency simulation of products and applications
› Fast parameter configuration with interpreter window
› Digital/analog co-simulation
› 430+ application circuits (lighting, power supplies, motor control, computing PoL)
Infineon Designer Use Cases: Online Analog-Digital Co-Simulation with Code Debugger

1. Select **XMC1200 circuit**

2. Choose simulation mode

3. Co-simulate MCU software with analog circuit

Example circuit: **32-bit MCU XMC1200 controlling the RGB color walk with constant brightness**
Infineon Designer Use Cases: Digital Twin
24V Arduino Shield PROFET™+ 24V Family

› Unique Value Proposition
  - Customer explore the board by “click & play”, w/o reading through manuals & datasheets
  - Customer adapt the soft board (hardware & software) to his own application needs prior to Buy Online

› Full Hardware & Software Design
  - HW: Arduino Shield
  - SW: DAVE
  - Online Circuit: TINA SPICE
  - Engine: DesignSoft
  - Hosting: Infineon & DesignSoft

Software

Hardware

Online Virtualization

1 SW Debugger

2 HW Oscilloscope
Agenda

1. Infineon Designer SPICE Simulation Overview
2. Infineon Designer New Features
3. Getting Started Examples
4. Support
Infineon Designer Highlights: Full-featured Circuit Editor

1. **myInfineon login**
2. **Viewer & circuit editor**

- Login with myInfineon account
- Create your circuit from scratch or based on existing Infineon example circuits
Infineon Designer Highlights:
Design Tool - Parameter Setting & Calculation

1. Wanna try it out? Click on analysis
2. Double click on green window to design
3. If you like what you see, buy online
4. Enjoy other circuits

Interpretter window

Design Tool

- Easier parameter setting
- Quicker calculation & circuit configuration with defined formulas
Infineon Designer Highlights: Additional Features

**Parametric configurable circuit**

1. Wanna try it out? Click on "Simulate Transient".
2. Set application parameters below or directly change any component.

**Power dissipation & efficiency**

**Enhanced MCU debugging**

**Signal processing, e.g. ripple**

Copyright © Infineon Technologies AG 2018. All rights reserved.
Agenda

1. Infineon Designer SPICE Simulation Overview
2. Infineon Designer New Features
3. Getting Started Examples
4. Support
Infineon Designer – SPICE Simulation Demo, Examples & Hands-on

› **Getting Started**
  - Learn about Infineon Designer basics
  - Simulate a simple inverse buck example
  - Edit your circuit in the schematic editor

› **Advanced simulation features with a boost circuit**
  - Simulate a boost example with CoolMOS P7 & CoolSiC Diode
  - Check startup / steady-state simulation, and it’s advanced features
  - Set parameters with Design Tool for circuit configuration
  - Hands-on: create your first Infineon Designer circuit

› **Motor Control with XMC/DAVE™ Software in the Loop**
  - Explore the functions of a real evaluation board
  - Run software together with hardware
  - Step through the code in the online debugger
  - Install DAVE™ software development suite, alter your code, and upload new code

› How to import a SPICE model
1. Open circuit **Getting Started**

2. Click on **Simulate Transient**

3. Learn about functions & signals in the Diagram Window

4. Click on the MOSFET symbol and click on “...” “SubCkt-Type”

5. Select a new MOSFET from the drop-down menu

6. Login with your myInfineon account to save your circuit
Infineon Designer Example

**Getting Started** (Part 2/2: circuit editing)

1. Open circuit **Getting Started** and Login with your myInfineon account
2. Select different tabs and display the parts from “Infineon” and other circuit elements like Basic (R, L, C), Meters, Sources, Semiconductors, Spice Macros, etc.
3. Click on Basic and add a capacitor and place it
4. Select and right click to rotate
5. Select the capacitor terminal and wire it
6. Select “Save as” from the menu “File” and save under “MyCircuits”
7. Click on **Simulate Transient**
Infineon Designer Example

**Boost Circuit** (Part 1/4: system efficiency)

1. Open circuit Boost Circuit with power types and Login with your myInfineon account
2. Open menu “Analysis -> Transient...” & make sure the displayed signals are in the steady-state, check “Power dissipation analysis enabled”
3. Click on Simulate Efficiency to check system efficiency and power losses
4. Select “Power dissipation” table, Evaluate efficiency, losses and component pass/fail
5. Select “Transient” diagram, zoom all and zoom in to check details of the signals, use cursor and to measure signal values
Infineon Designer Example  
Boost Circuit  (Part 2/4: signal ripple)

1. Within the same diagram window, open Menu -> Process -> Ripple…, open ripple calculation window

2. Select the curves for ripple calculation, e.g.: V_out, I_out

3. Check relative and absolute ripple values for individual curves

4. Use the similar procedure for other signal processing possibilities, e.g. averaging, Fourier analysis, etc.
Infineon Designer Example
**Boost Circuit** (Part 3/4: Design Tool)

1. Close the signal diagram window

2. Double click the green box to open Design Tool for circuit parameter configuration, after configuration click “Run” to execute the settings

3. Circuit components with changed values will be highlighted in red

4. Open Menu -> Tools -> Design Tool... to configure the parameter settings and limited ranges
**Infineon Designer Example**

**Boost Circuit** (Part 4/4: Design Tool)

1. Write basic functions for value assignment of every component to be configurable, e.g. \( V_{\text{in}} := V_{\text{IN}} \), \( V_{\text{in}} \) is the component label name, \( V_{\text{IN}} \) is a global variable to be used to set the component \( V_{\text{in}} \) in the Design Tool, formulas are allowed for calculation.

2. Add new parameter, e.g. \( V_{\text{IN}} \), set value, min. max. and give comment

3. Redundant parameters can be removed

4. Click “Run” to execute the configuration of the Design Tool

5. Click “OK” to confirm and close the window

6. Click menu bar symbol and click “Insert Design Text…” to place it in the circuit

7. Save circuit & run simulation
Infineon Designer XMC Software Example
H-Bridge Kit 2Go (Part 1/3: circuit debugging)

1. Open Design **H-Bridge Kit 2Go**
2. Click on **Simulate Transient**
3. Learn about functions & signals in the Diagram Window
4. Enable MCU Code Debugger
5. Start Simulation in interactive mode and step through the code
6. Purchase Sample & Board

---

Copyright © Infineon Technologies AG 2019. All rights reserved. Infineon Proprietary
Infineon Designer XMC Software Example

**H-Bridge Kit 2Go** (Part 2/3: software upload)

1. Open Design **H-Bridge Kit 2Go** and Login with MyInfineon Account
2. Download code .zip file **H-Bridge Kit 2Go – Default Simple Example Routine** example from the H-Bridge Kit board page Software & Tools tab
3. Click on the red XMC1100 symbol and open the MCU code window by clicking on “…”
4. Browse to the .zip file and select it (contains files .elf, .hex and source files)
5. Upload the code HBK_2Go Simple Motor Control_DAVE4.zip-SW-v01_00-EN.zip
6. Click on **Simulate Transient**
7. Check the results
Infineon Designer XMC™ software example
H-bridge Kit 2Go (Part 3/3: software rebuild)

1. Download and install DAVE™ software development suite
2. Download code .zip file H-bridge Kit 2Go – default Simple example routine from the H-bridge kit board page
3. Start DAVE™ and import the unzipped project into your workspace with file → import → Infineon DAVE™ project
4. Click on next and browse to the folder with the project and select it
5. Change the code and rebuild
6. Go to your workspace directory and .zip the updated project folder
7. Got back to the schematic click on the red XMC1100 symbol and upload the MCU code
How to import a SPICE model (Part 1/2)

1. **Login** with your myInfineon account
2. Select File -> New, create a new circuit
3. Click menu bar symbol `+` and click “Upload macro…”
4. Name your model, select “from file”, and browse to your SPICE model in .SUBCKT format
5. Click on “Upload”

Example Model **OrCAD Capture for IR2110**
6. Edit symbol pin layout
7. Optional: Replace “?” with new pin name in symbol
   Example: VDD/? -> VDD/VDD
8. Optional: place pins on top, left, right, bottom
9. Click on OK to upload macro
10. Click menu bar symbol and click “Insert macro…”
11. Select IR2110 macro and place it on your schematic
12. Finalize your circuit and Save it with “Save -> Save as”
13. Test your circuit
Agenda

1. Infineon Designer SPICE Simulation Overview
2. Infineon Designer New Features
3. Getting Started Examples
4. Support
Technical Support

Please visit www.infineon.com/support

Support Page
Support is available in English, German and Mandarin from our talented team of experts.

Find an answer to your question

Please state your question (with at least 3 words)

FAQ
1. Technical Support [CN] [DE]
2. Chip Card and Security Dists [CN] [DE]
3. HiRel Discretes for special applications, e.g. Aero and Space [CN] [DE]
4. Supplier Service, Supplier Page, page registration [CN] [DE]
5. Use Infineon Designer for Simulation and Development of your Circuit [CN] [DE]
6. How to login to myinfineon [CN] [DE]
Resource List  https://www.infineon.com/tools

Finder Tools
›  Infineon Solution Finder
›  Infineon Product Finder

Hardware Simulation Tools

Thermal design
›  Infineon IPOSIM Power Simulation for Power Modules and Disk Devices
›  Infineon Power Simulation for Integrated Power Modules (IPM) powered by PLECS
›  Infineon Power Simulation for discrete IGBTs powered by PLECS

Electrical & software design
›  Infineon Designer powered by TINACloud
›  PowerEsim Simulation for Switched-Mode Power Supply (SMPS)

Magnetic design
›  Infineon Magnetic Sensor Design Tools

Software Development Tools

›  DAVE™ Development Platform for XMC™ 32-bit Industrial Microcontroller based on ARM® Cortex®-M
›  TriCore™ Development Tools for AURIX™ 32-bit Automotive Microcontroller based on TriCore™

Infineon Support & Distribution Partners
›  Technical Assistance Center
›  Forums
›  Newsletter
›  Orderable Part Number (OPN) Finder
›  Where to Buy your Products
›  Contact & Locations
Disclaimer

The information given in this training materials is given as a hint for the implementation of the Infineon Technologies component only and shall not be regarded as any description or warranty of a certain functionality, condition or quality of the Infineon Technologies component.

Infineon Technologies hereby disclaims any and all warranties and liabilities of any kind (including without limitation warranties of non-infringement of intellectual property rights of any third party) with respect to any and all information given in this training material.
Part of your life. Part of tomorrow.