Quartus II 软件使用教程

Cytech-XA

Vincent Song

Q2 2008

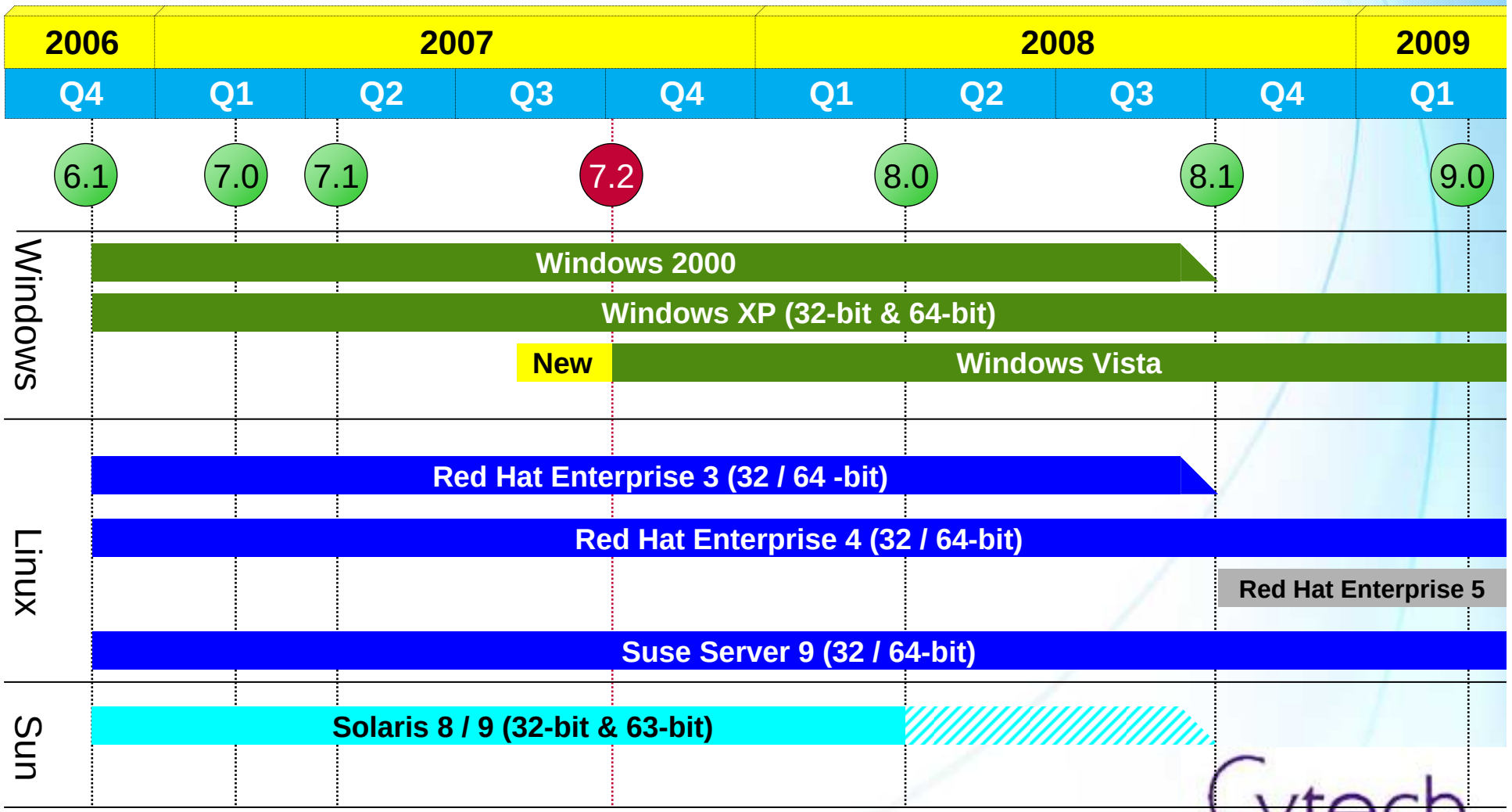
Programmable Logic Families

- Structured ASIC
  - HardCopy® II & HardCopy Stratix
- High & medium density FPGAs
  - Stratix III, Stratix II & Stratix
- Low-cost FPGAs
  - Cyclone III, Cyclone II & Cyclone
- FPGAs w/ clock data recovery
  - Stratix II GX & Stratix
- Low-cost 90-nm FPGAs for PCI Express, Gigabit Ethernet,and Serial RapidIO up to 2.5 Gbps
  - Arria GX

CPLDs

  - MAX II, MAX 7000 & MAX 3000
- Configuration devices
  - Serial (EPCS) & enhanced (EPC)

QuartusII软件发布RoadMap



Cytech Technology

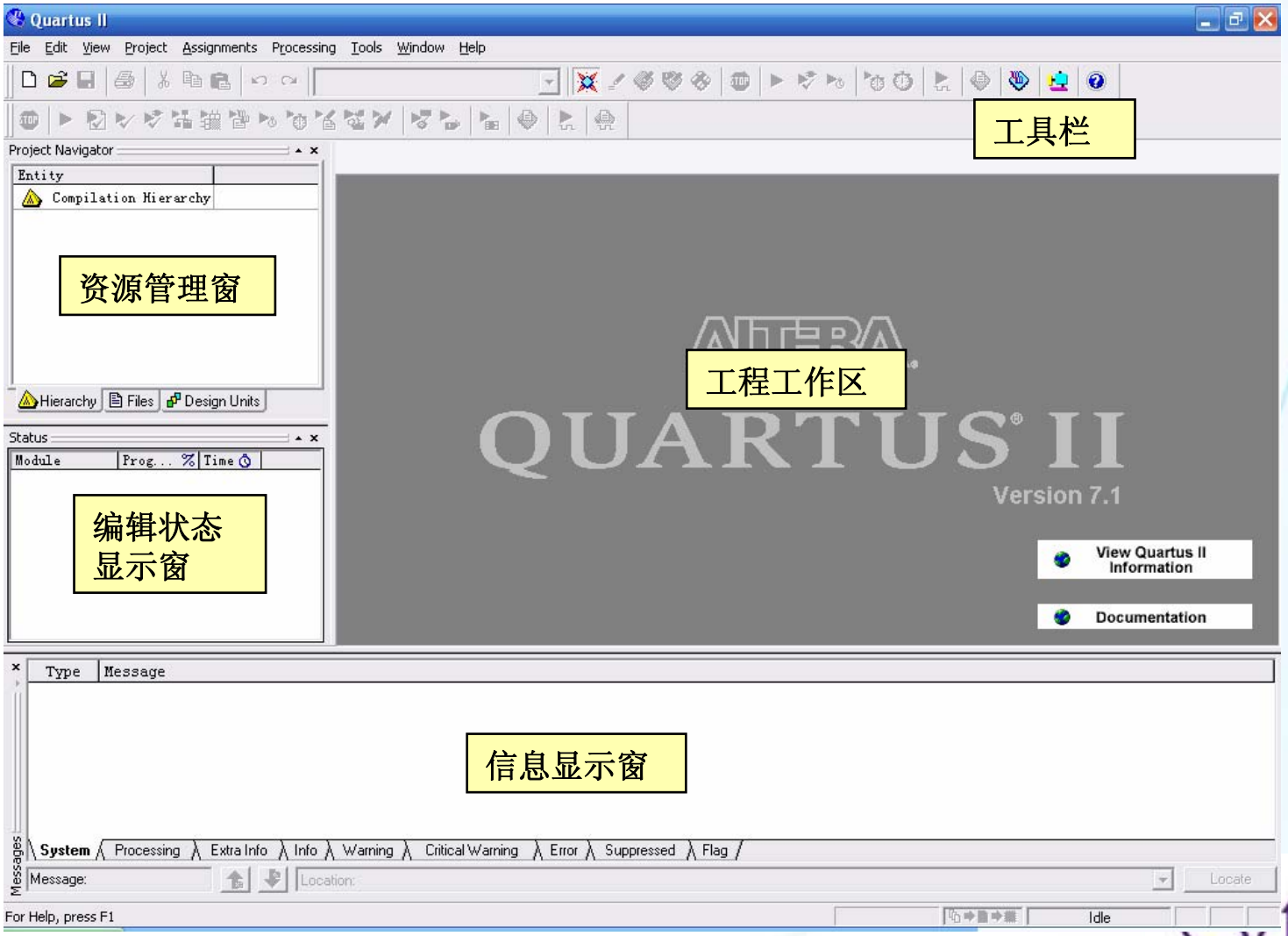
从QII 6.1开始支持多核处理器和64位OS

- Multi-processor cores now mainstream
  - Benefit → faster compile times

- 64-Bit O/S – moving mainstream
  - Benefit → access to more than 2 GB of memory

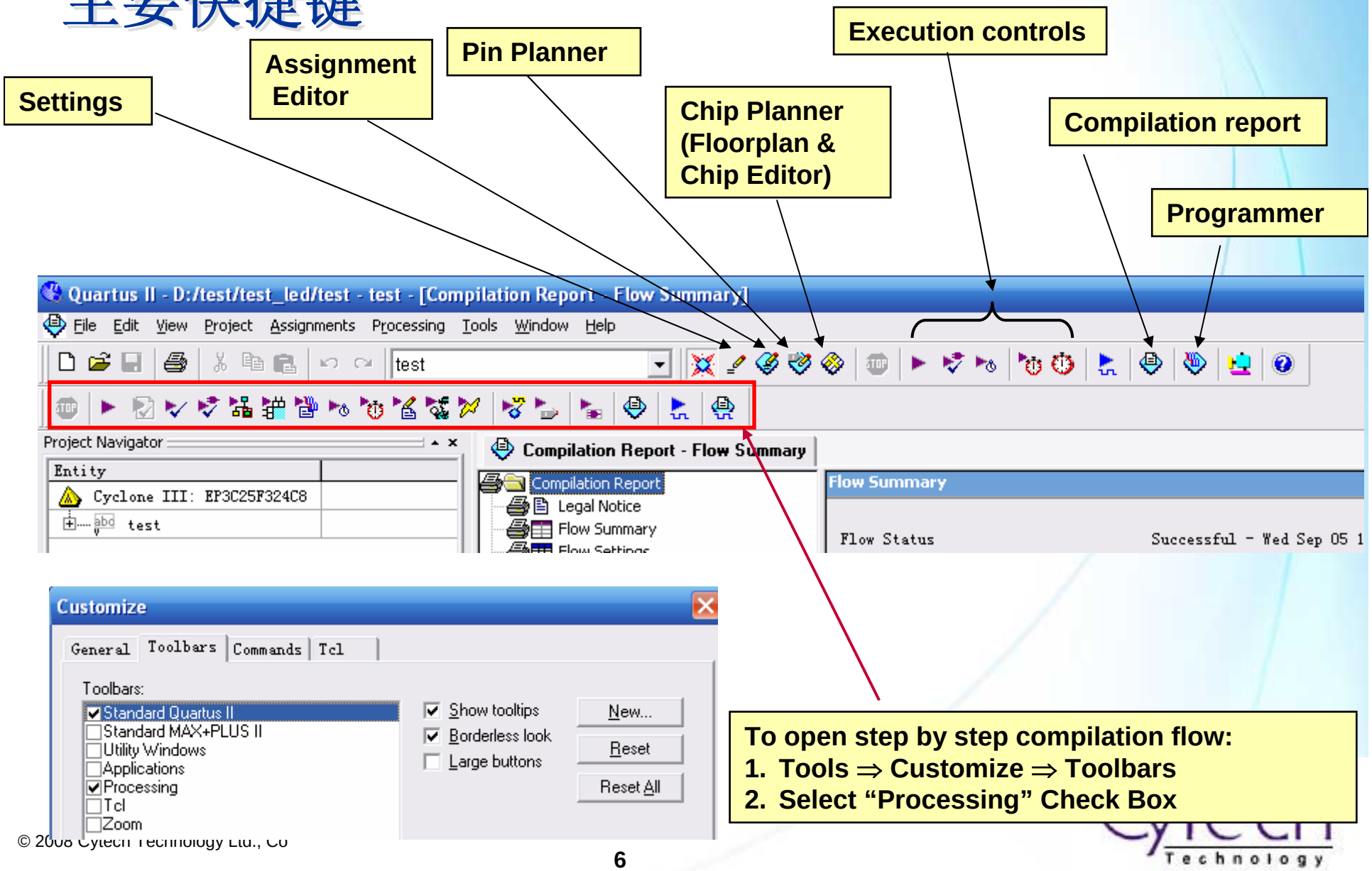
Cytech Technology

Quartus II开发环境



Cytech Technology

主要快捷键



To open step by step compilation flow:
1. **Tools ⇒ Customize ⇒ Toolbars**
2. **Select “Processing” Check Box**

Agenda

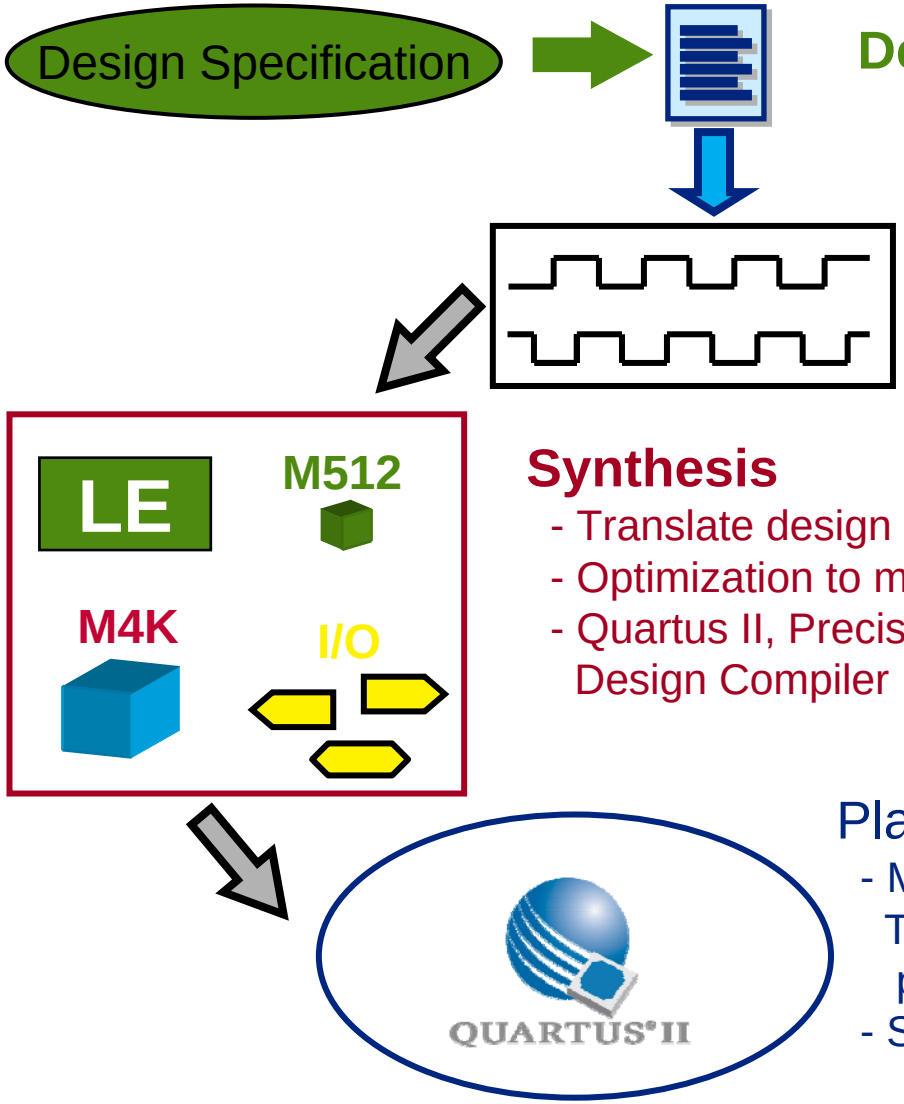
- 设计流程概要
- 建立工程
- 设计输入
- 编译
- 综合
- 使用Synplify Pro做综合
- 布局布线
- Assignment Editor
- 管脚分配
- 仿真
- 器件编程
- 时序约束
- SignalTap II 逻辑分析器仪

Cytech Technology

Quartus II 软件使用教程

设计流程概要

Typical PLD Design Flow



Design entry/RTL coding

- Behavioral or structural description of design

RTL simulation

- Functional simulation (ModelSim®, Quartus II)
- Verify logic model & data flow (no timing delays)

Synthesis

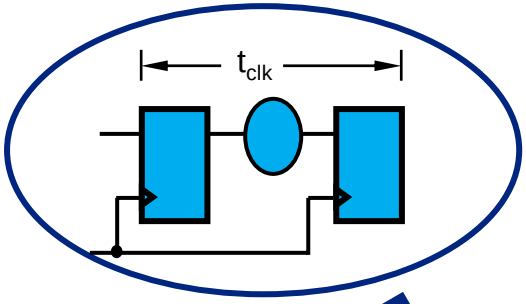
- Translate design into device specific primitives
- Optimization to meet required area & performance constraints
- Quartus II, Precision Synthesis, Synplify/Synplify Pro, Design Compiler FPGA

Place & route

- Map primitives to specific locations inside Target technology with reference to area & performance constraints
- Specify routing resources to be used

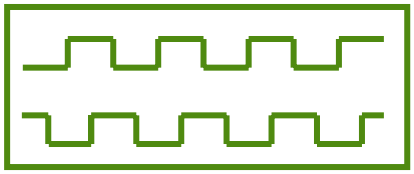
Cytech Technology

Typical PLD Design Flow



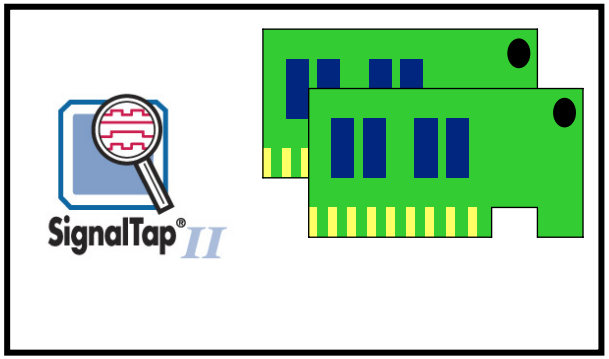
Timing analysis

- Verify performance specifications were met
- Static timing analysis



Gate level simulation

- Timing simulation
- Verify design will work in target technology



PC board simulation & test

- Simulate board design
- Program & test device on board
- Use **SignalTap II** for debugging

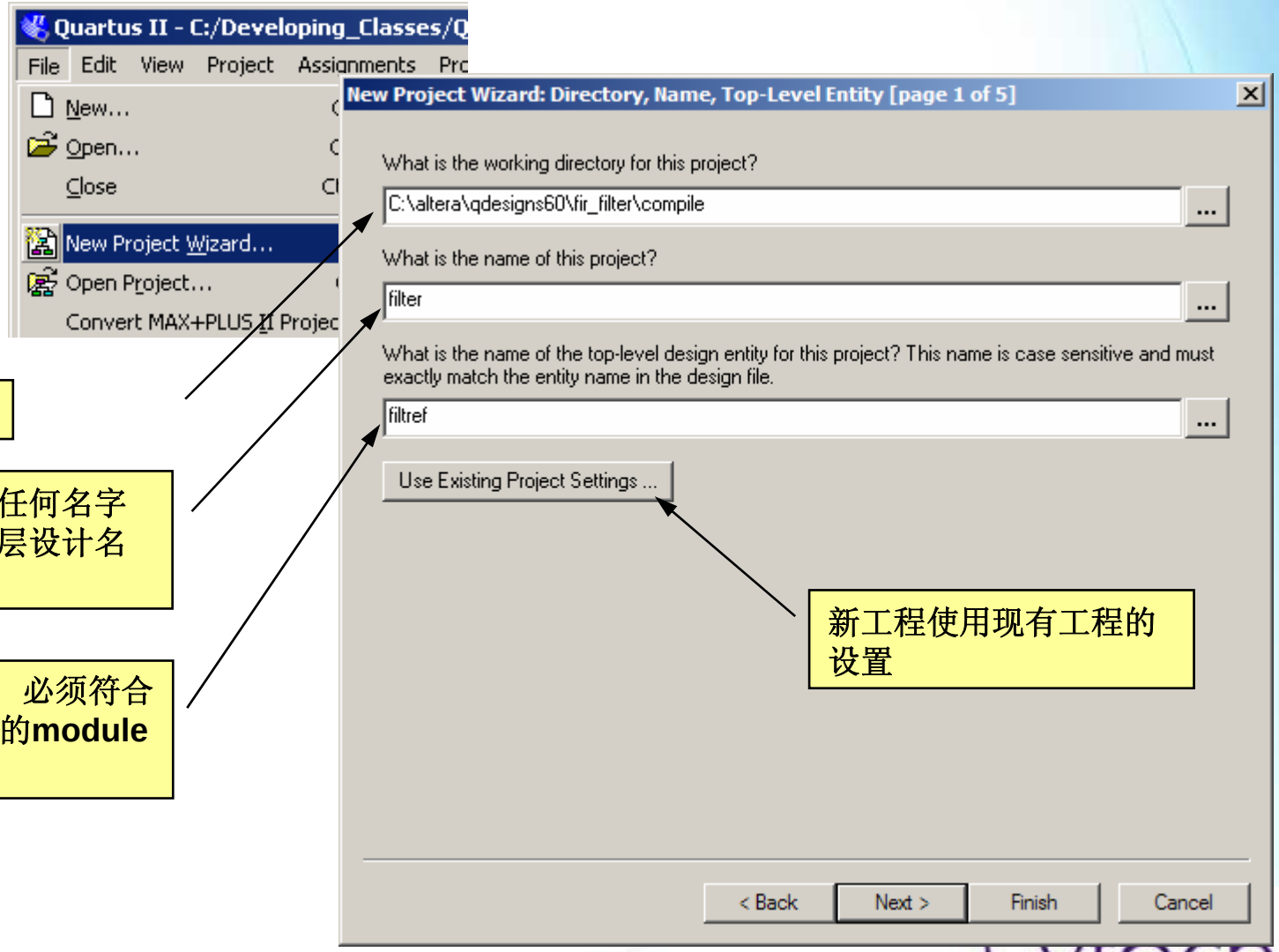
Cytech Technology

Quartus II 软件使用教程

建立工程

设计新工程使用New Project Wizard比较方便

File menu



选择工程的路径

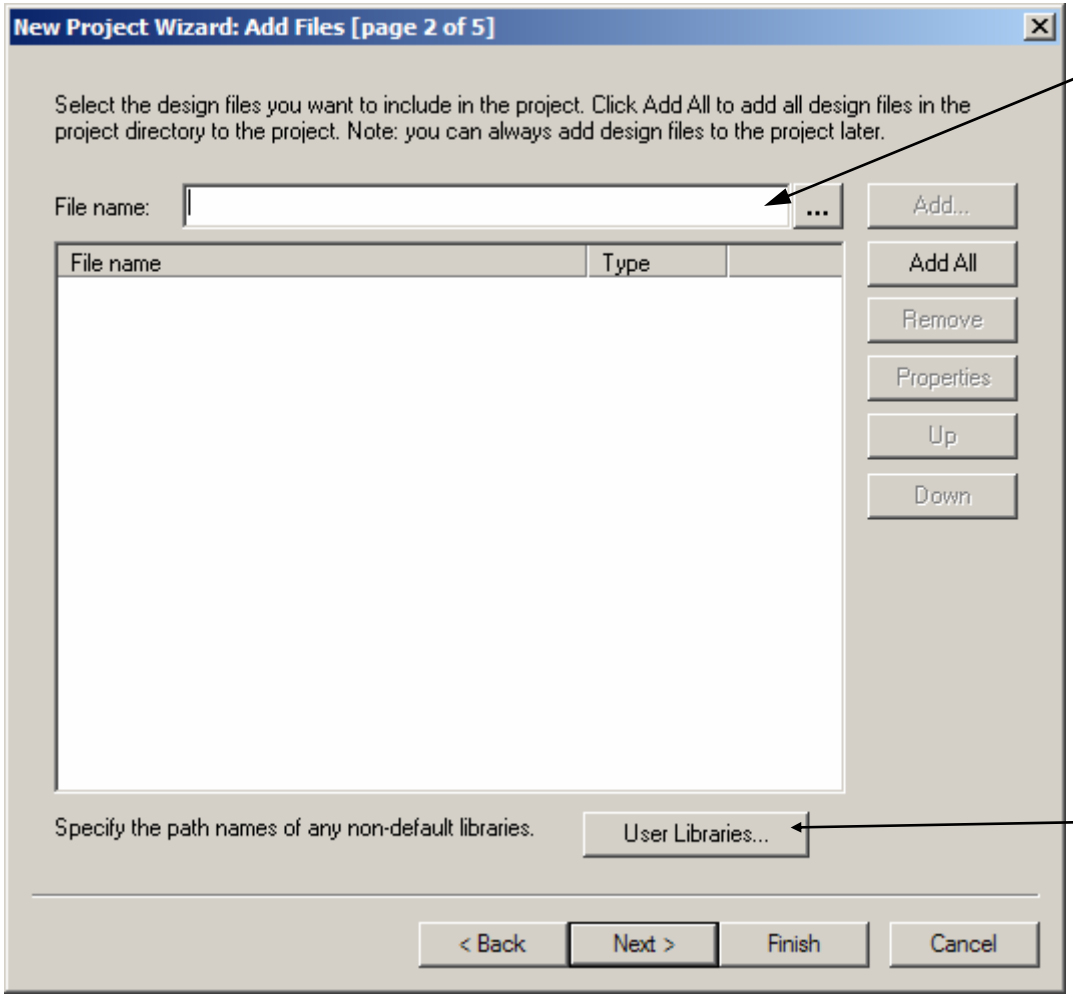
工程名可以使用任何名字，建议使用和顶层设计名相同的名字

顶层Entity名称，必须符合TOP文件中定义的module名称

新工程使用现有工程的设置

Cytech Technology

添加源文件（这一步骤可以跳过）



Add design files

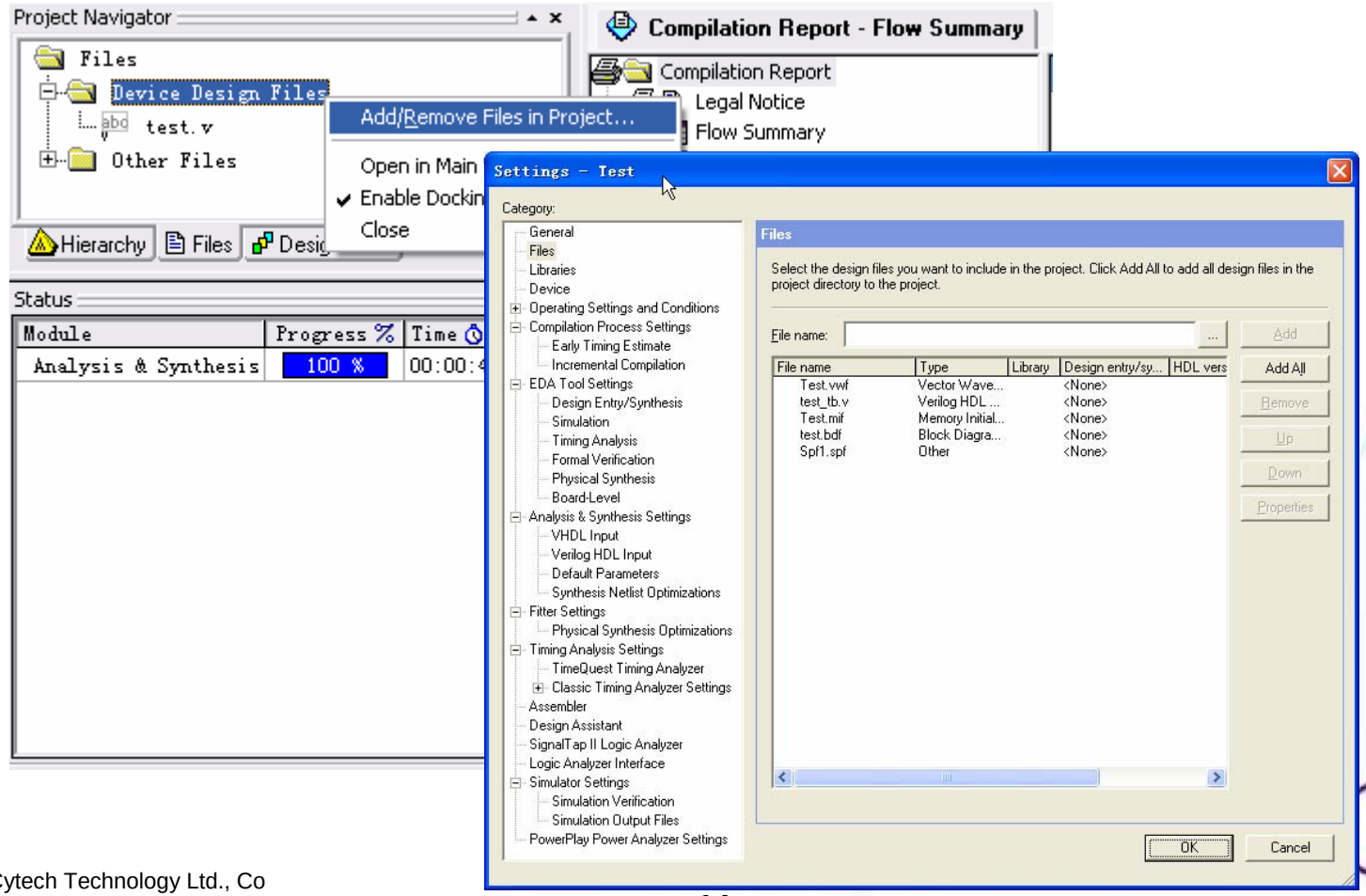
- **Graphic (.BDF, .GDF)**
- **AHDL**
- **VHDL**
- **Verilog**
- **EDIF**
- **VQM**

Add user library pathnames

- **User libraries**
- **MegaCore®/AMPP℠ libraries**
- **Pre-compiled VHDL packages**

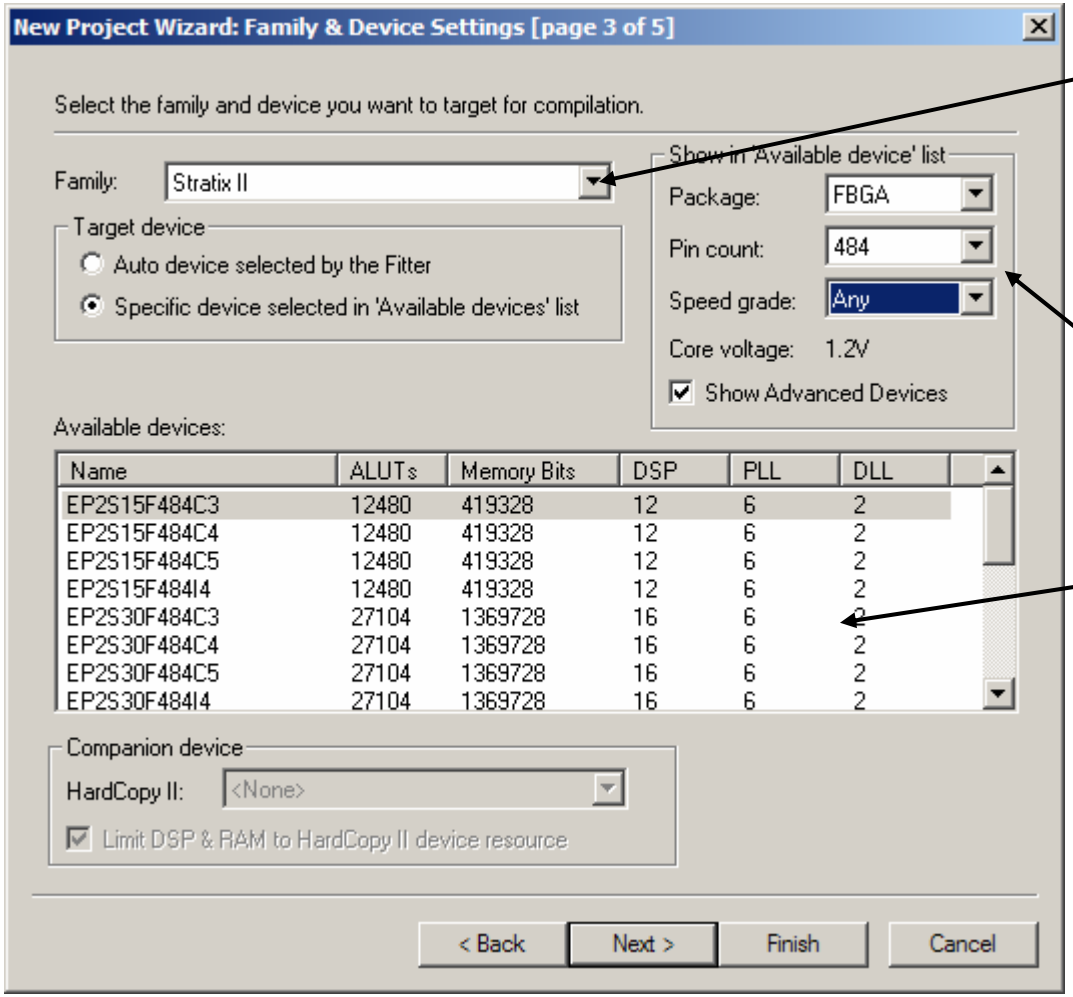
Cytech Technology

如果跳过新建向导的Add File，可以在工程生产完毕之后，在导航界面的File下点击“Device Design Files”，右键弹出菜单选择“Add/Remove Files in Project”



Cytech Technology

选择器件



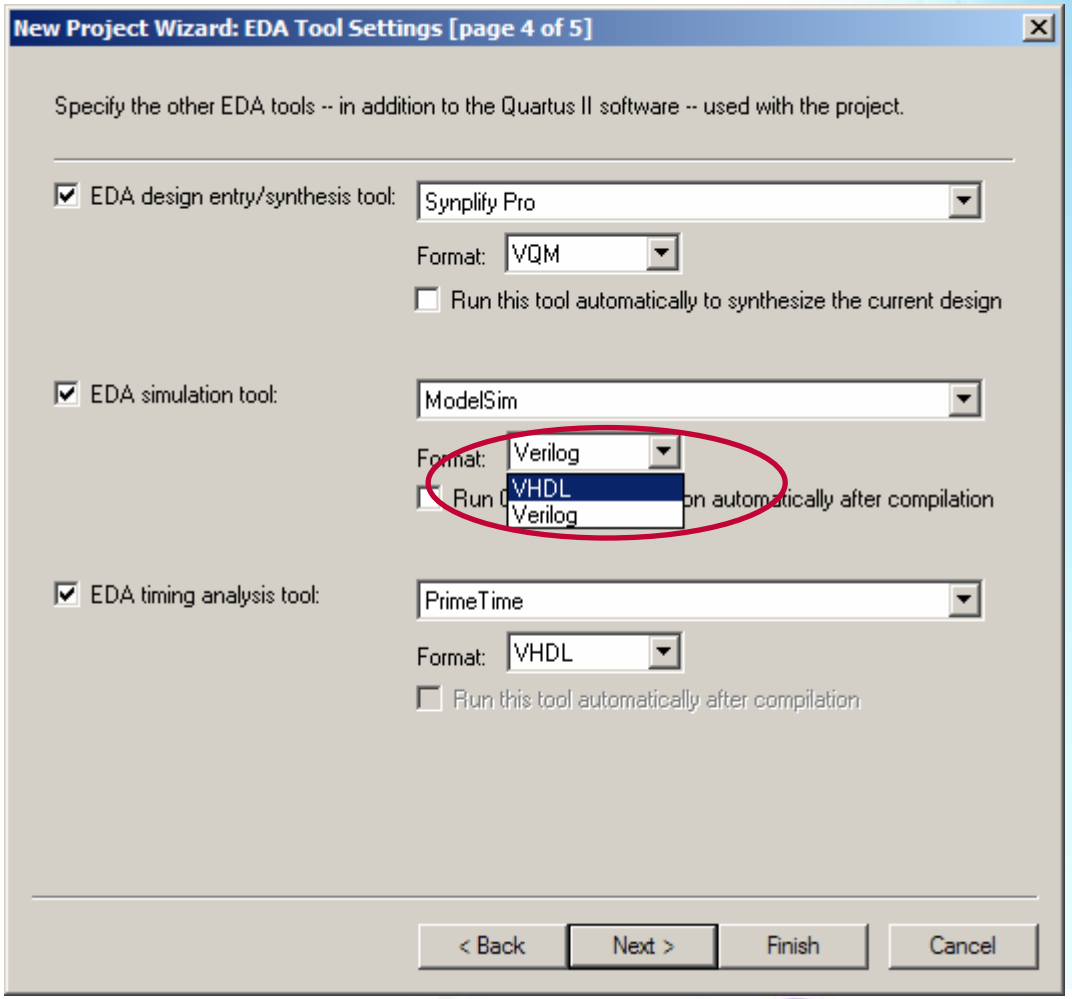
选择器件系列

Package可以选择器件的封装，**Pin count**可以选择器件的引脚数，**Speed grade**可以选择器件的速度等级，这些选项可以缩小可用器件列表的范围，以便快速找到需要的目标器件。

Cytech Technology

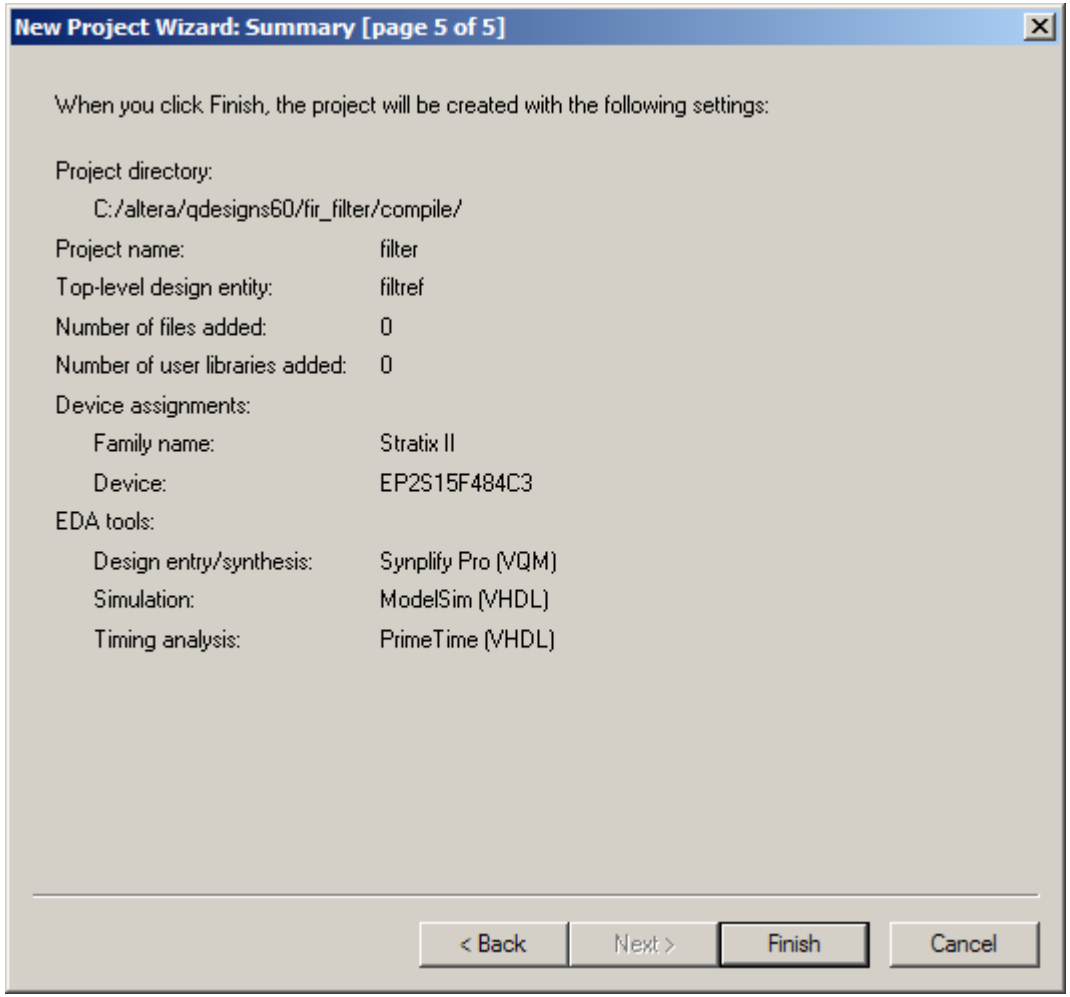
EDA 工具设置

选择综合、仿真、时序分析等第三方工具



Cytech Technology

完成!

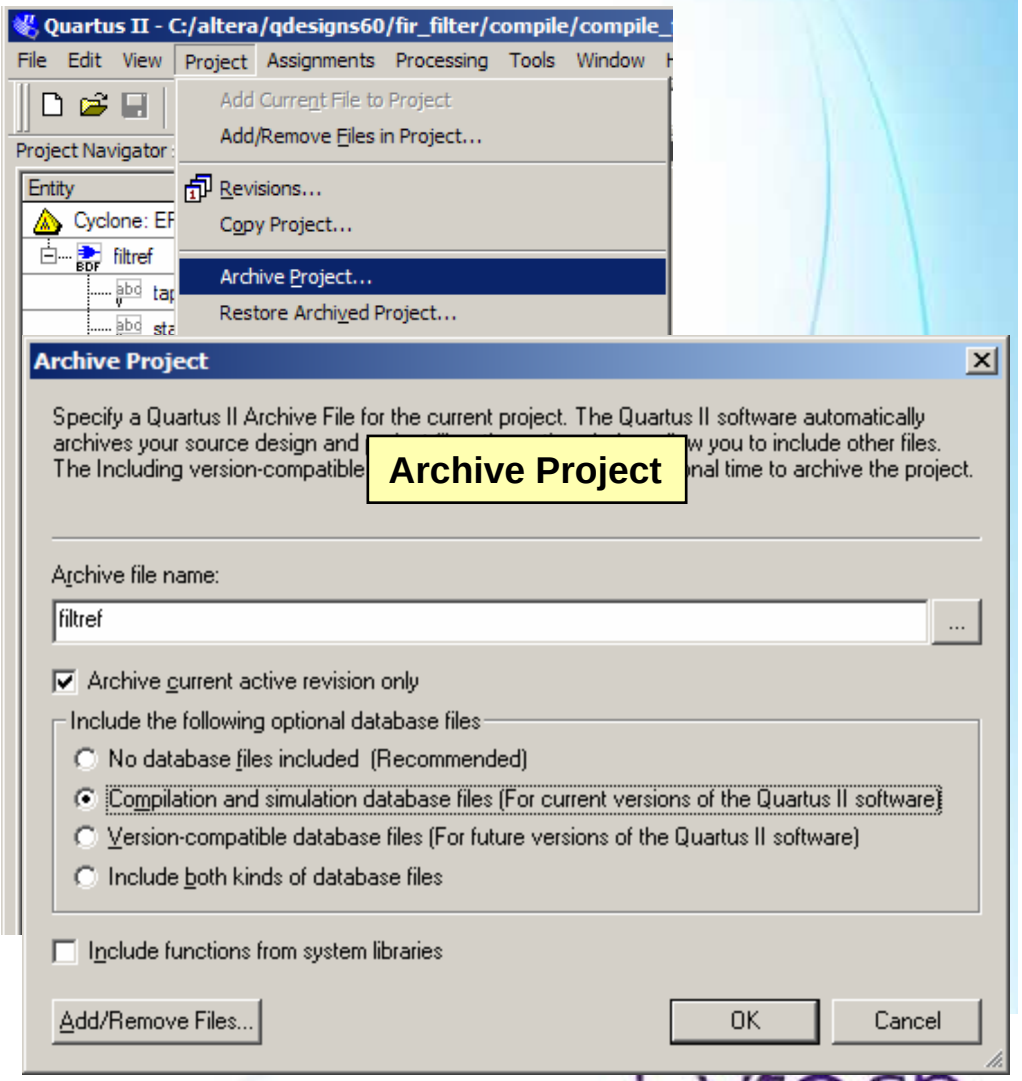


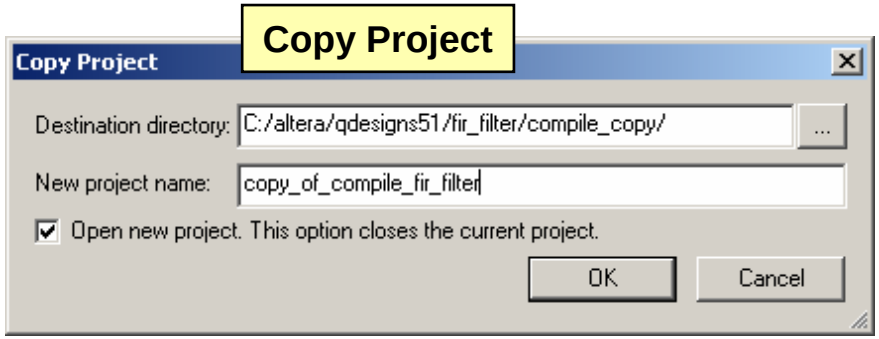
确认全部参数设置，若无误则单击**Finish**按钮，完成工程的创建；若有误，可单击**Back**按钮返回，重新设置。

Cytech Technology

工程管理

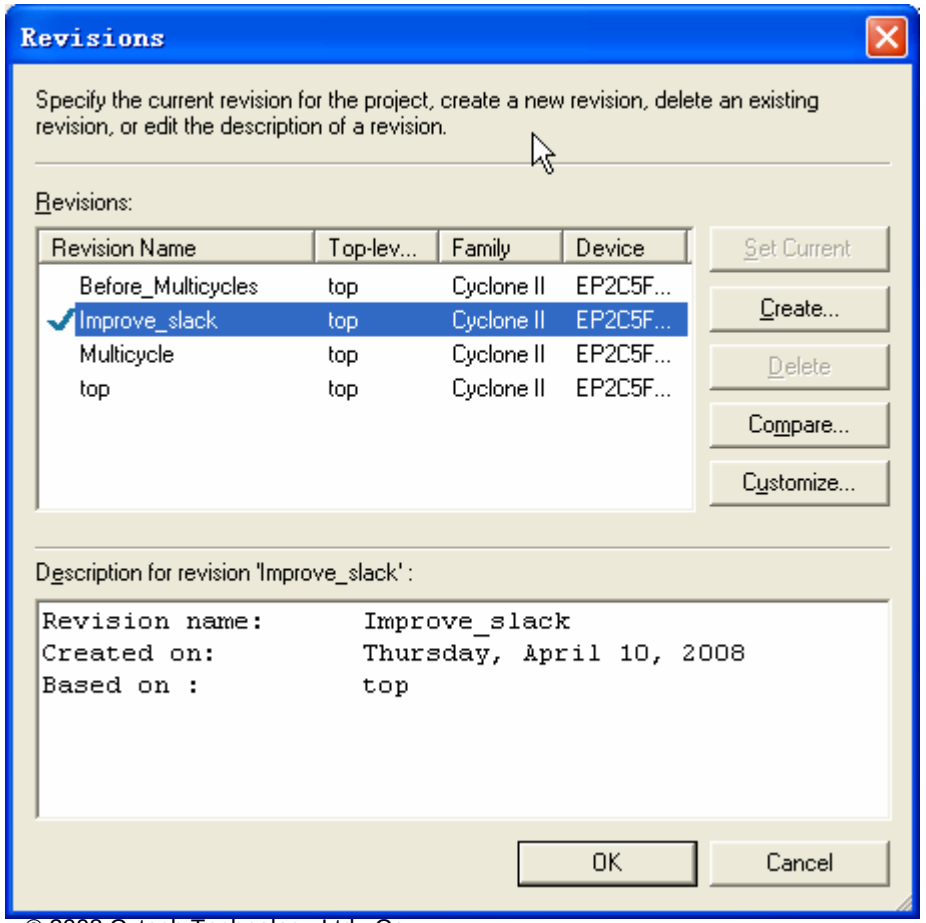
- 工程打包
  - 生成.qar文件
- 工程复制

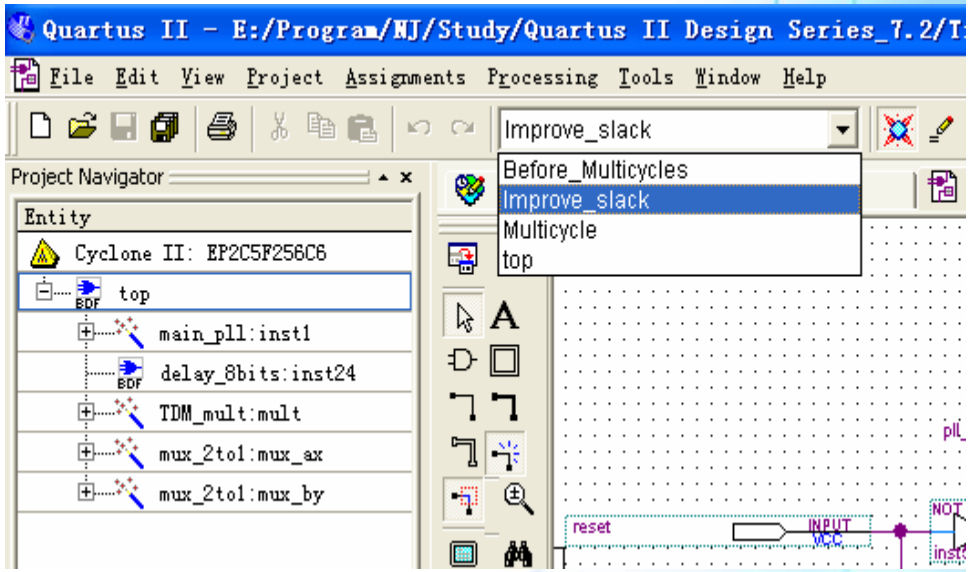




版本管理

- 通过菜单Project -> Revisions打开版本管理窗口，可以在原工程的基础上建立多个版本，并且可以比较，方便设计。
- 注意：不同的版本只能对约束做更改，如果更改原设计，则所有版本均会更改。



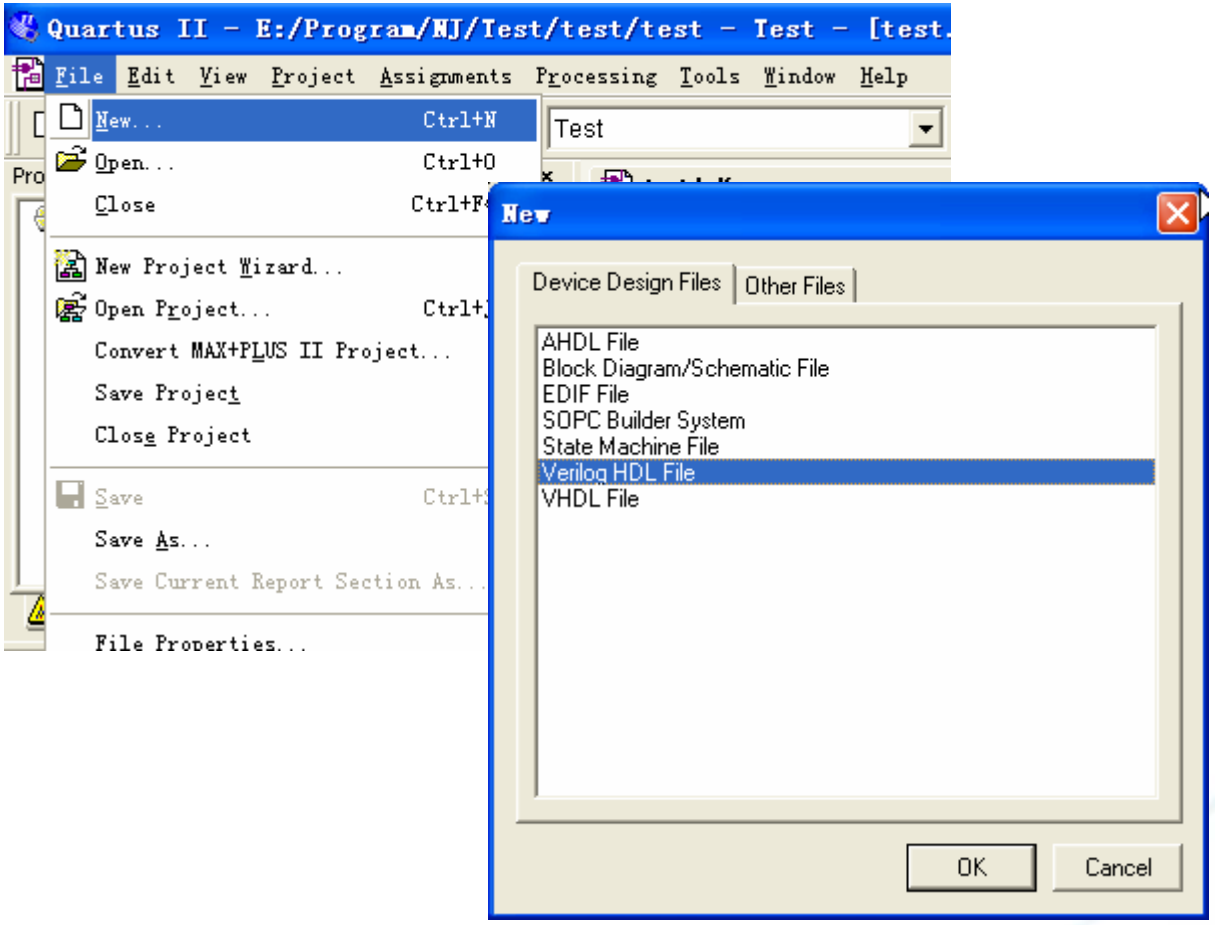


Cytech Technology

Quartus II 软件使用教程

设计输入

新建一个设计文件



选择要创建的文件类型

Cytech Technology

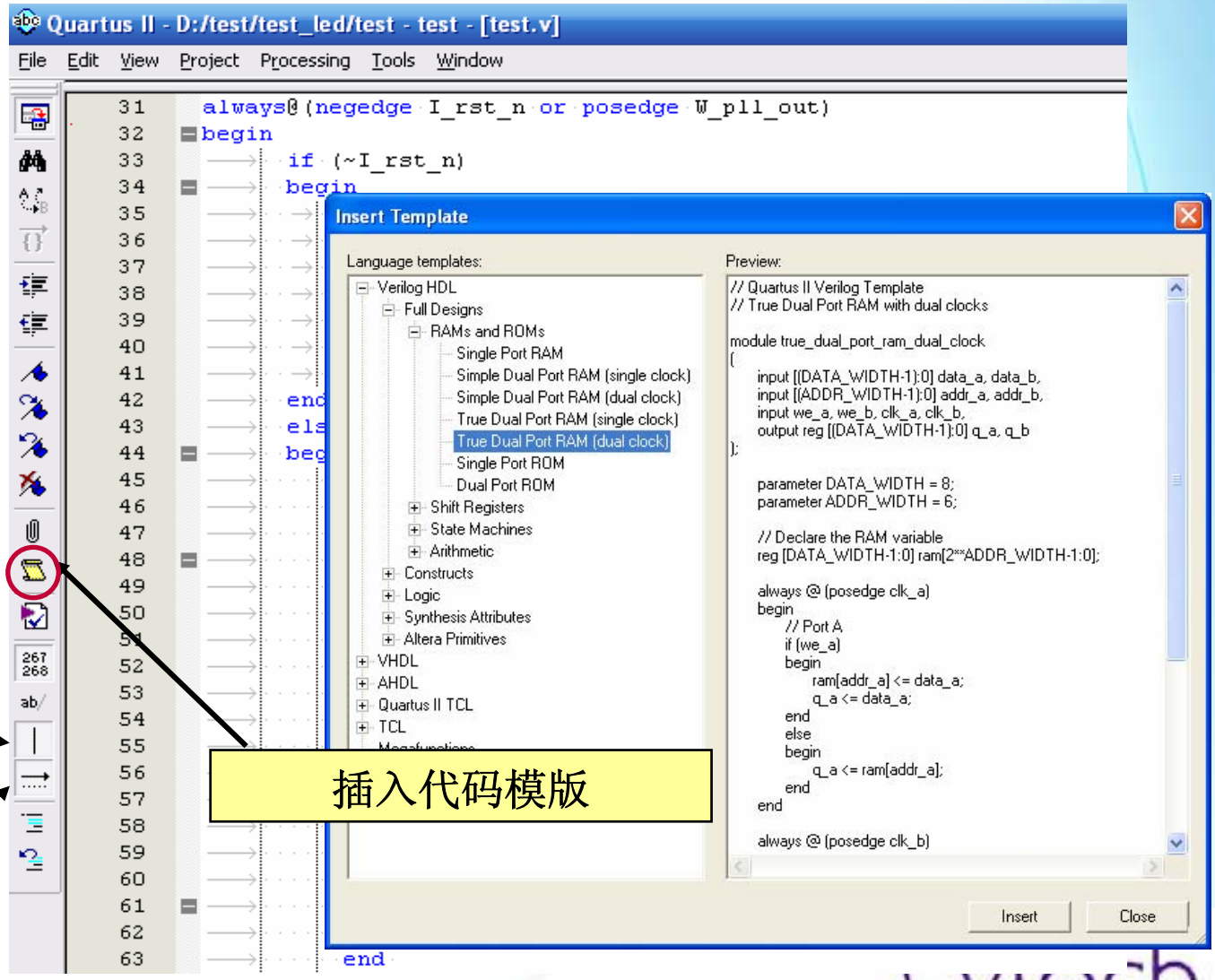
QII7.1文本编辑器

独立/整合窗口切换

“Alt”键实现列操作的切换

列对齐显示标记

行对齐显示标记

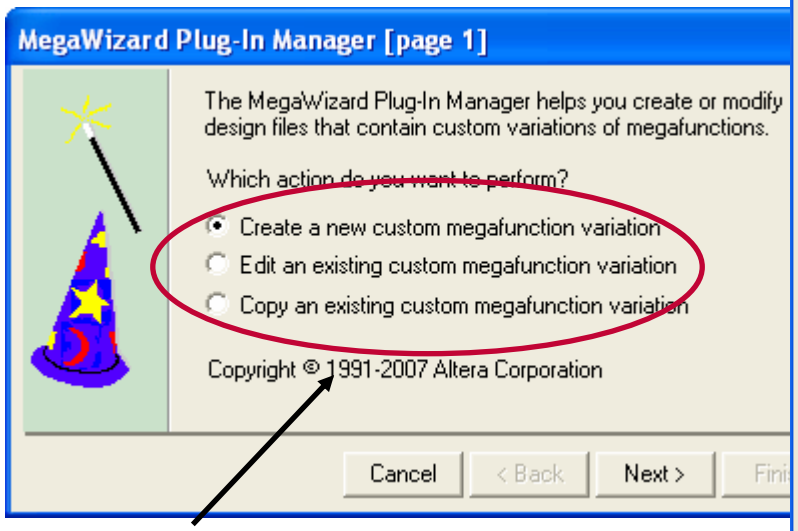


插入代码模版

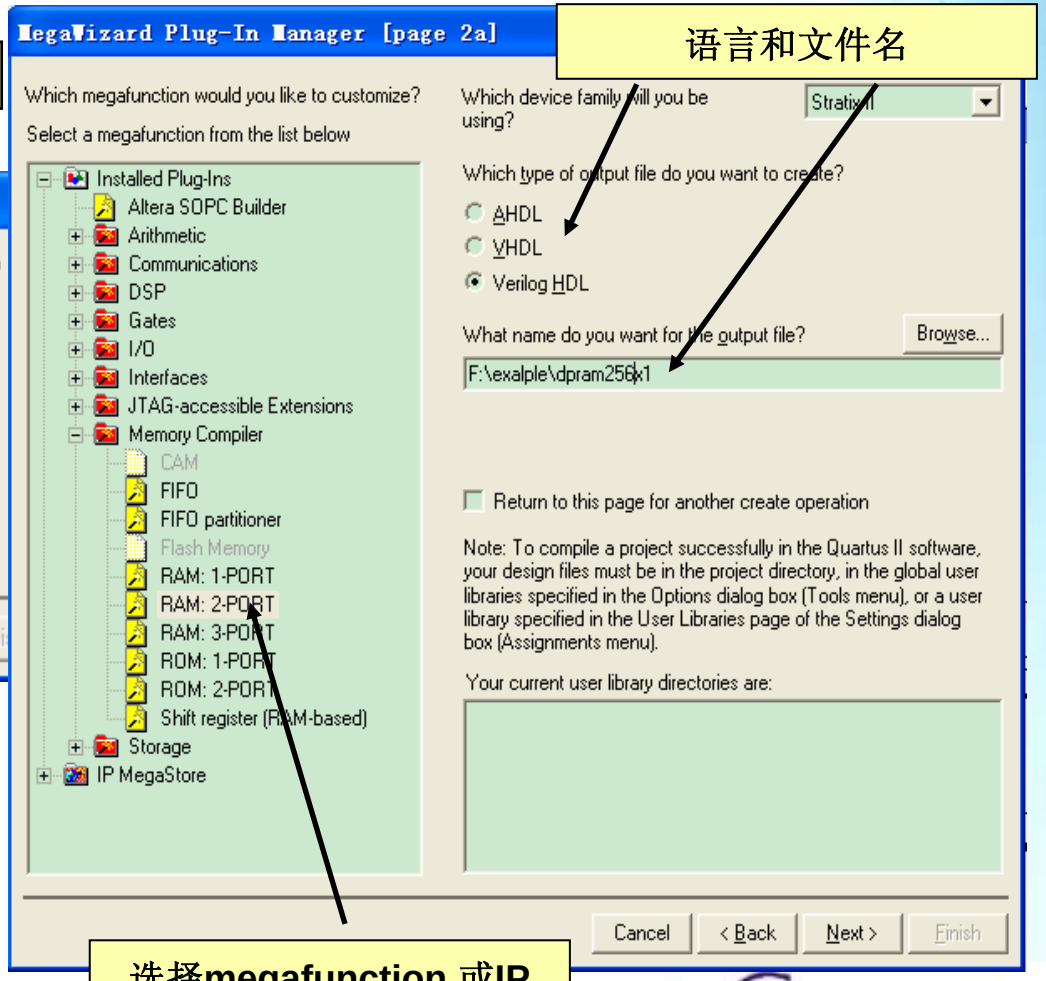
Cytech Technology

使用MegaWizard Plug-in Manager调用宏功能模块

Tools ⇒ MegaWizard Plug-In Manager



可以创建一个新的**IP**文件，也可以编辑已有的**IP**文件，或者拷贝已创建的文件。

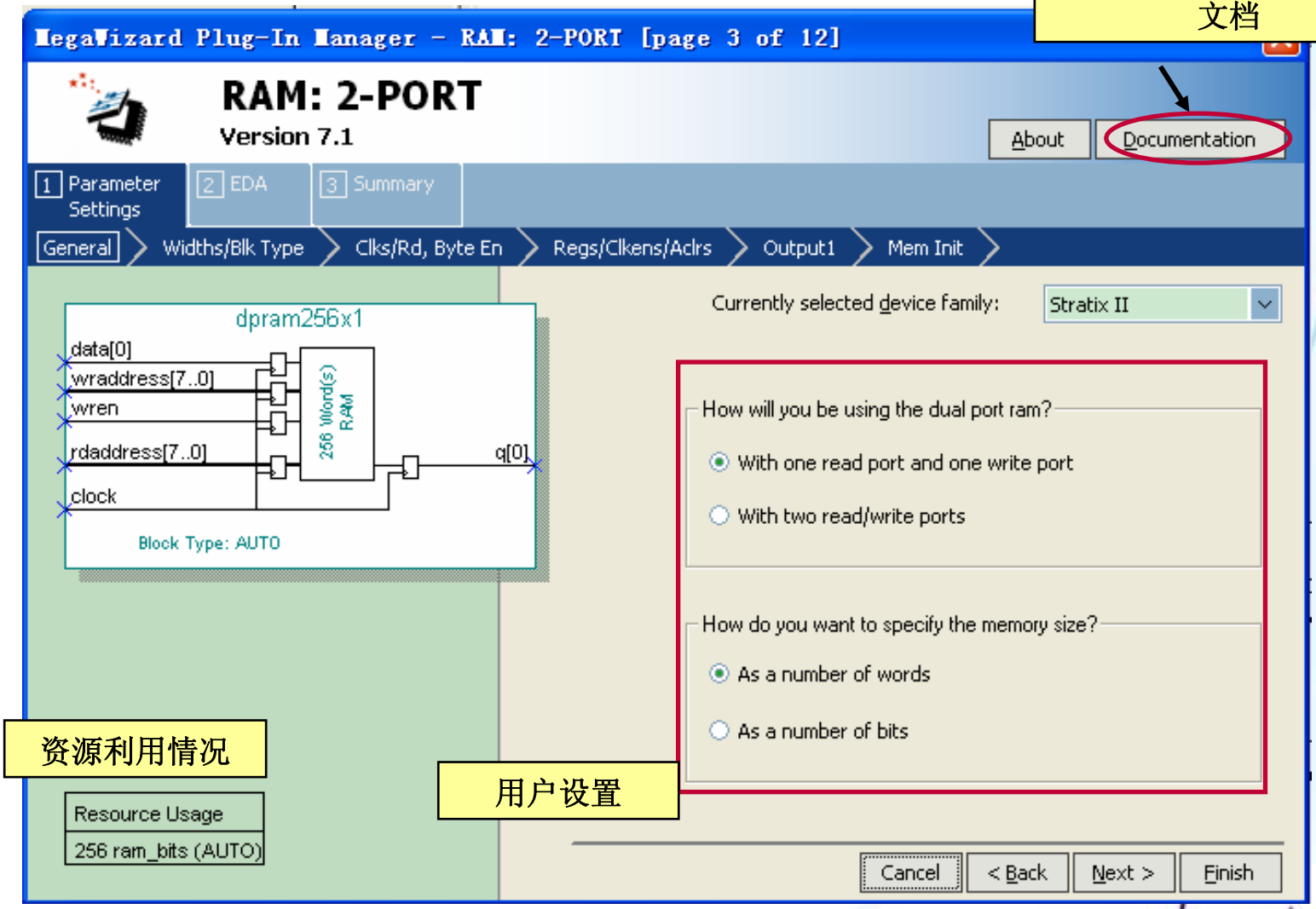


语言和文件名

选择**megafunction** 或**IP**

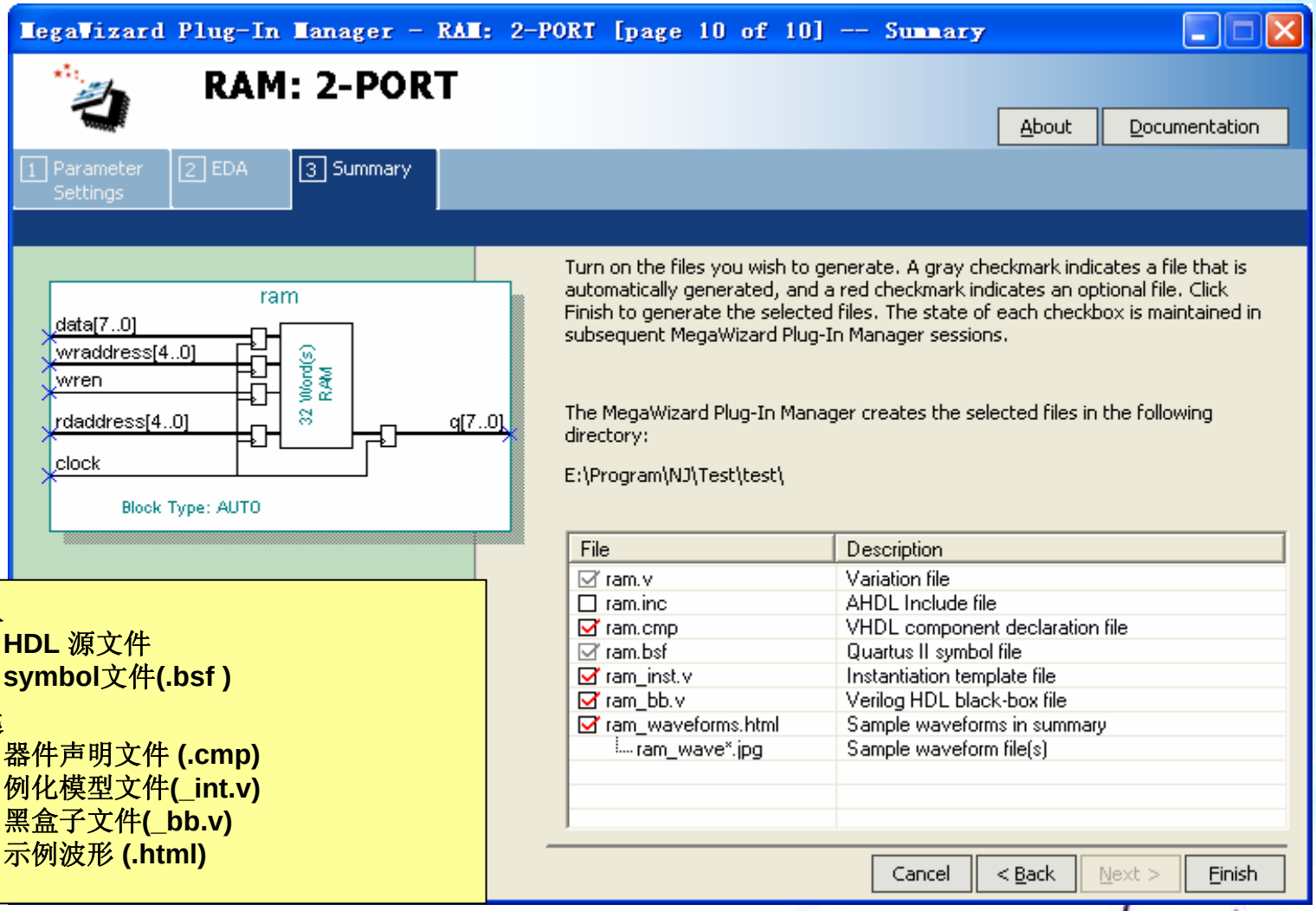
Cytech Technology

MegaWizard示例



Cytech Technology

MegaWizard示例



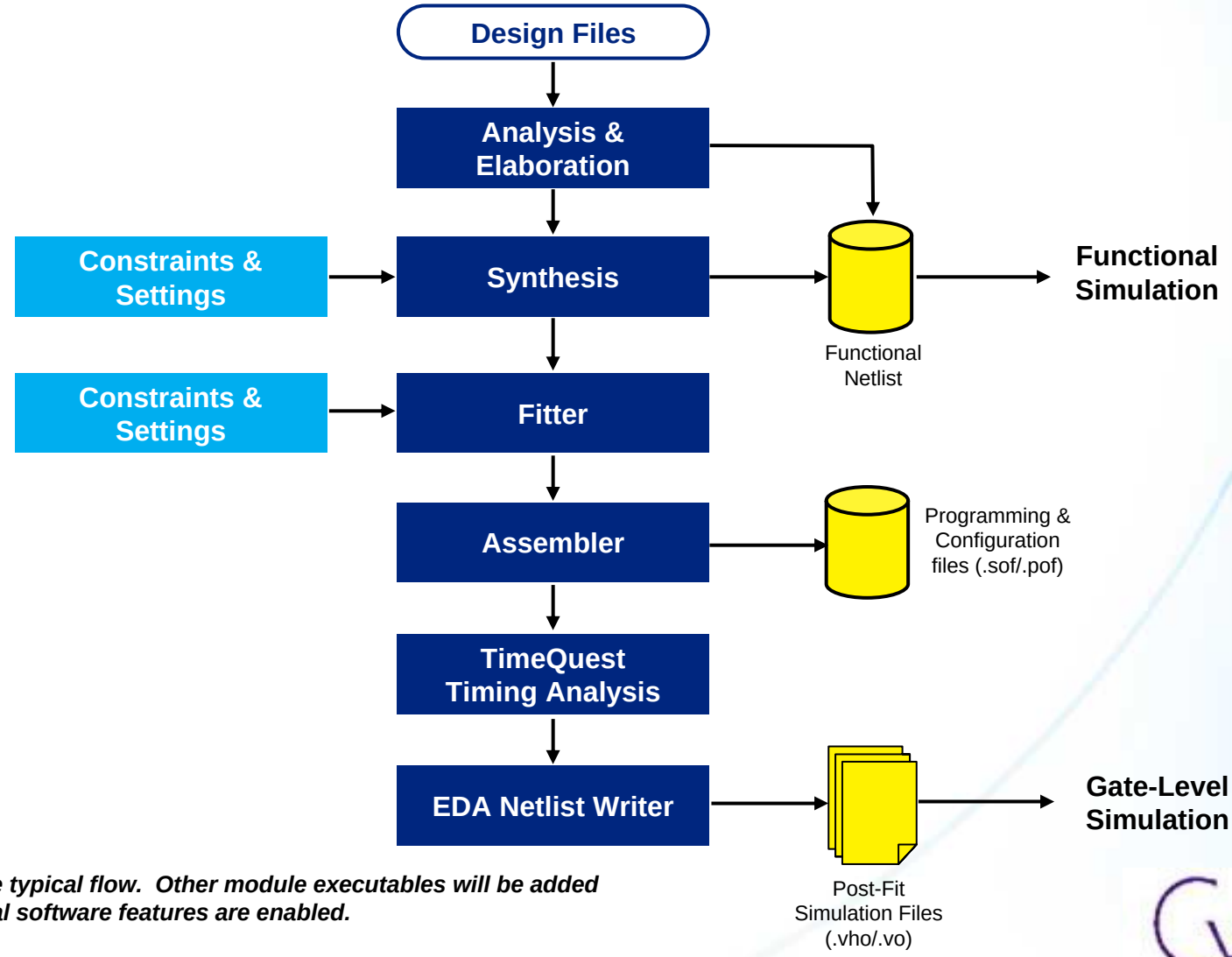
- 默认
  - **HDL 源文件**
  - **symbol文件(.bsf)**
- 可选
  - **器件声明文件 (.cmp)**
  - **例化模型文件(_int.v)**
  - **黑盒子文件(_bb.v)**
  - **示例波形 (.html)**

Cytech Technology

Quartus II 软件使用教程

编译

Qusrtus II全编译流程

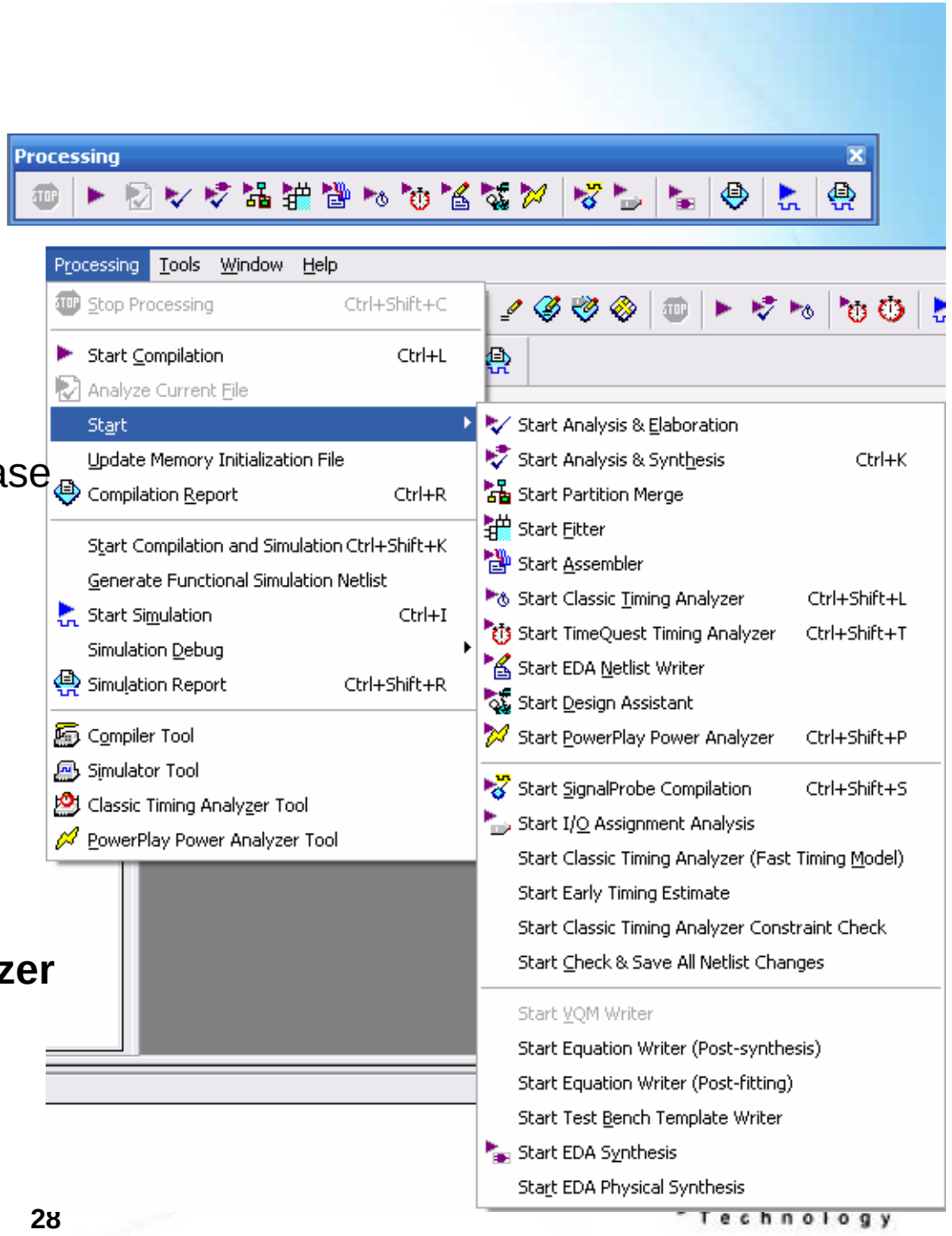


This is the typical flow. Other module executables will be added if additional software features are enabled.

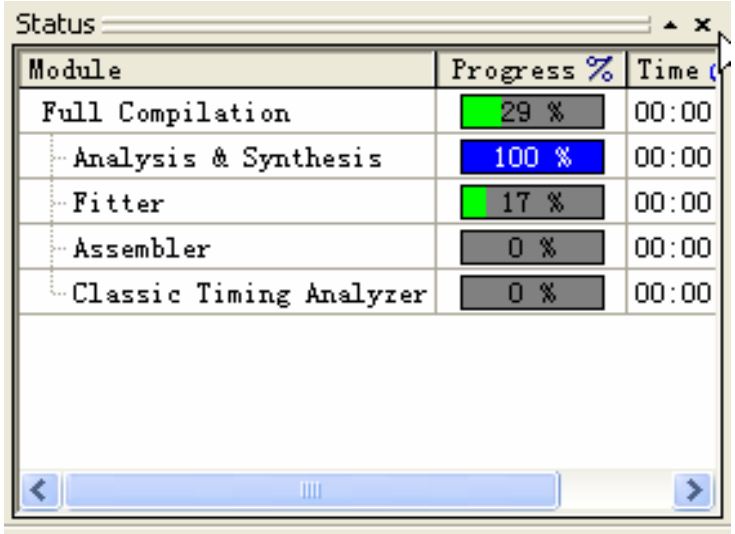
Cytech Technology

Processing选项

- **Start Compilation**
  - Performs full compilation
- **Start Analysis & Elaboration**
  - Checks syntax & builds database only
  - Performs initial synthesis
- **Start Analysis & Synthesis**
  - Synthesizes & optimizes code
- **Start Fitter**
  - Places & routes design
  - Generates output netlists
- **Start Assembler**
  - Generate programming files
- **Start TimeQuest Timing Analyzer**
- **Start I/O Assignment Analysis**
- **Start Design Assistant**



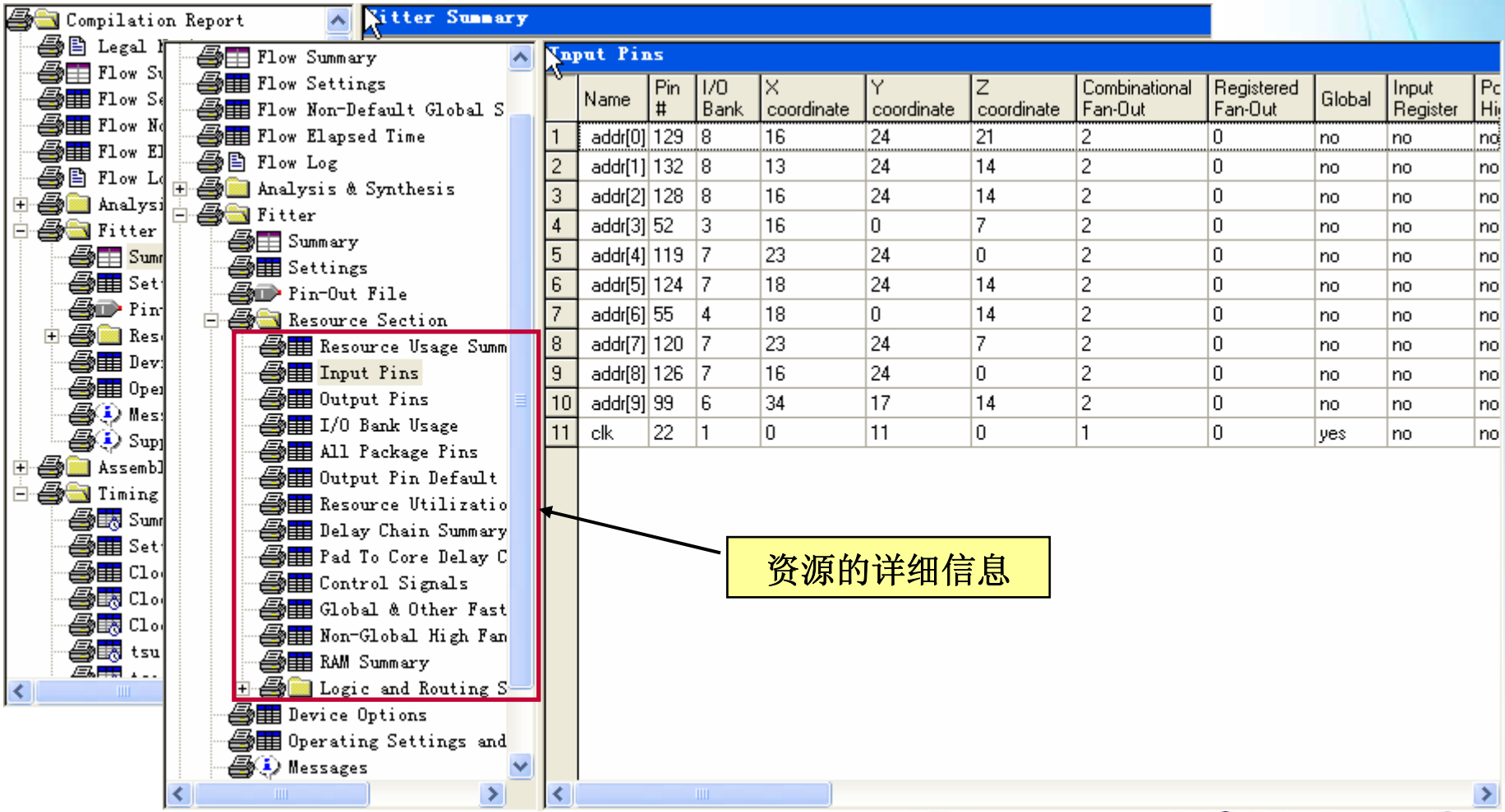
Status & Message Windows



- Analysis & Synthesis完成综合的功能
- Fitter是对设计进行布局布线
- Assembler为编程或配置目标器件建立一个或多个编程文件，包括.sof和.pof。
- Timing Analyzer作为全编译的一部分自动运行，它观察和报告时序信息，例如:：建立时间、保持时间 、时钟至输出延时 、引脚至引脚延时 、最大时钟频率 、延缓时间以及设计的其它时序特性。

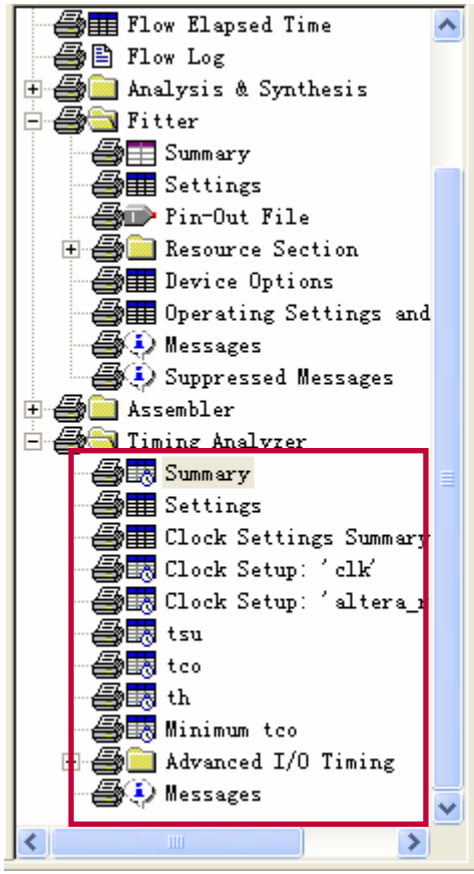
Cytech Technology

编译报告-资源报告



	Name	Pin #	I/O Bank	X coordinate	Y coordinate	Z coordinate	Combinational Fan-Out	Registered Fan-Out	Global	Input Register	Pc Hi
1	addr[0]	129	8	16	24	21	2	0	no	no	no
2	addr[1]	132	8	13	24	14	2	0	no	no	no
3	addr[2]	128	8	16	24	14	2	0	no	no	no
4	addr[3]	52	3	16	0	7	2	0	no	no	no
5	addr[4]	119	7	23	24	0	2	0	no	no	no
6	addr[5]	124	7	18	24	14	2	0	no	no	no
7	addr[6]	55	4	18	0	14	2	0	no	no	no
8	addr[7]	120	7	23	24	7	2	0	no	no	no
9	addr[8]	126	7	16	24	0	2	0	no	no	no
10	addr[9]	99	6	34	17	14	2	0	no	no	no
11	clk	22	1	0	11	0	1	0	yes	no	no

编译报告-时序报告



Timing Analyzer Summary

	Type	Slack	Required Time	Actual Time
1	Worst-case tsu	N/A	None	6.384 ns
2	Worst-case tco	N/A	None	12.451 ns
3	Worst-case th	N/A	None	-1.217 ns
4	Worst-case Minimum tco	N/A	None	7.534 ns
5	Clock Setup: 'altera_reserved_tck'	N/A	None	173.55 MHz (period = 5.762 ns)
6	Clock Setup: 'clk'	N/A	None	Restricted to 180.05 MHz (period = 5.554 ns)
7	Total number of failed paths			

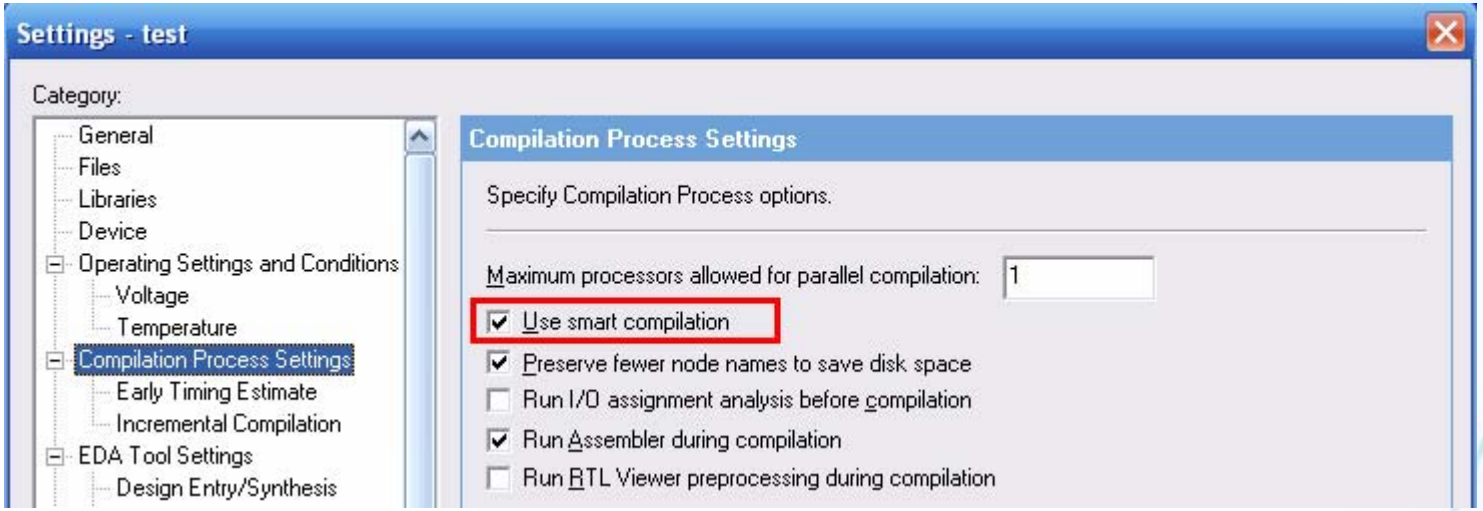
- 时序报告中按时序要求由差至好排列
- 报告中首列一般为Slack值
- Slack = Largest Required Time - Longest Actual Time
  - -Slack为正值，表示符合时序要求，显示为黑色
  - -Slack为负值，表示不符合时序要求，显示为红色

Cytech Technology

Quartus II 软件使用教程

综合

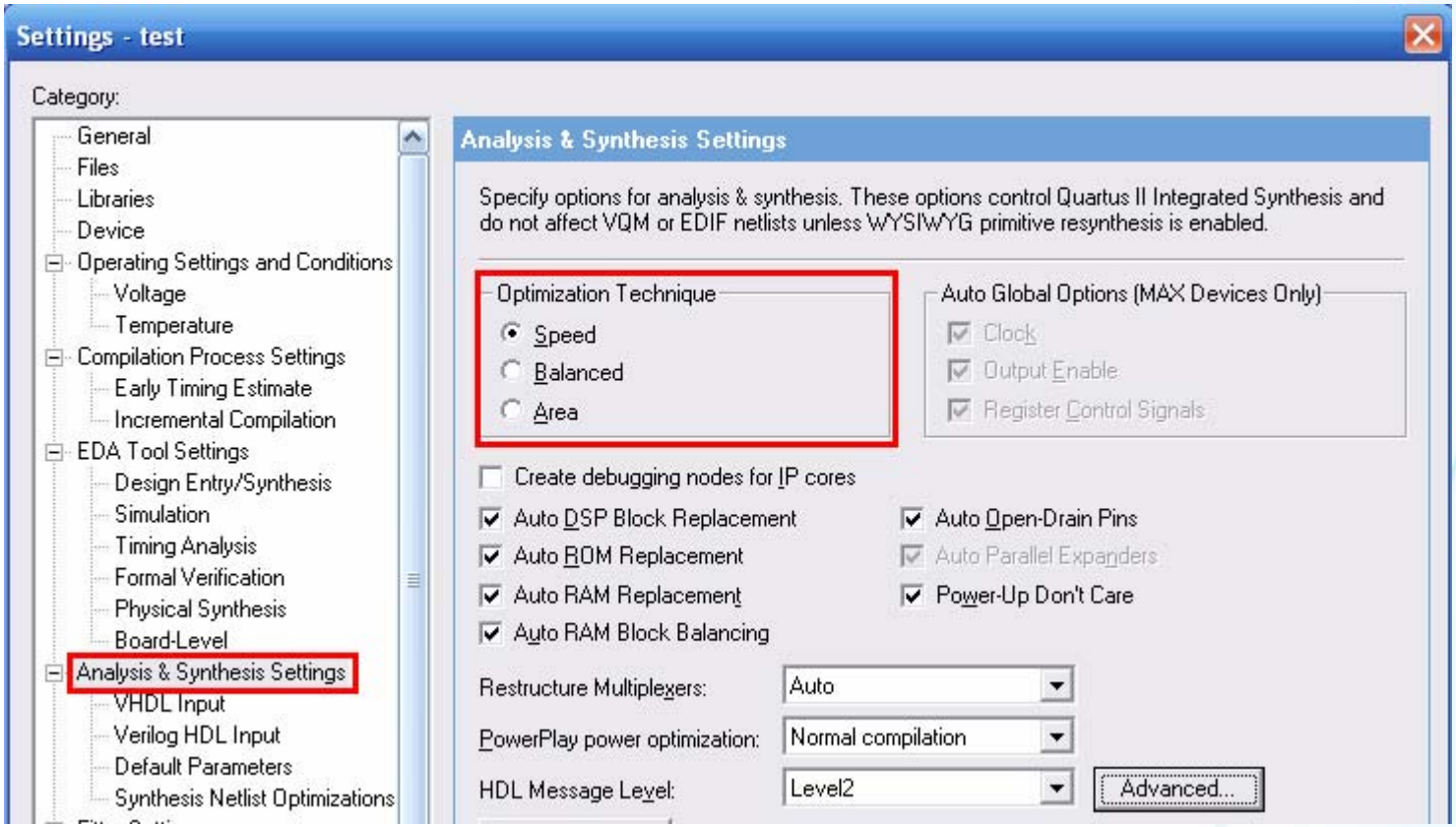
与Synthesis相关的设置(1)



- 如果选中，可以减少工程的编译时间。比如在工程中没有改动源代码而只是对约束进行了修改，使用了Smart Compilation 选项，则进行全编译（ ▶ 按钮）过程中，软件会自动跳过“Analysis & Synthesis”步骤。
- **QII默认设置为关闭，建议打开。**

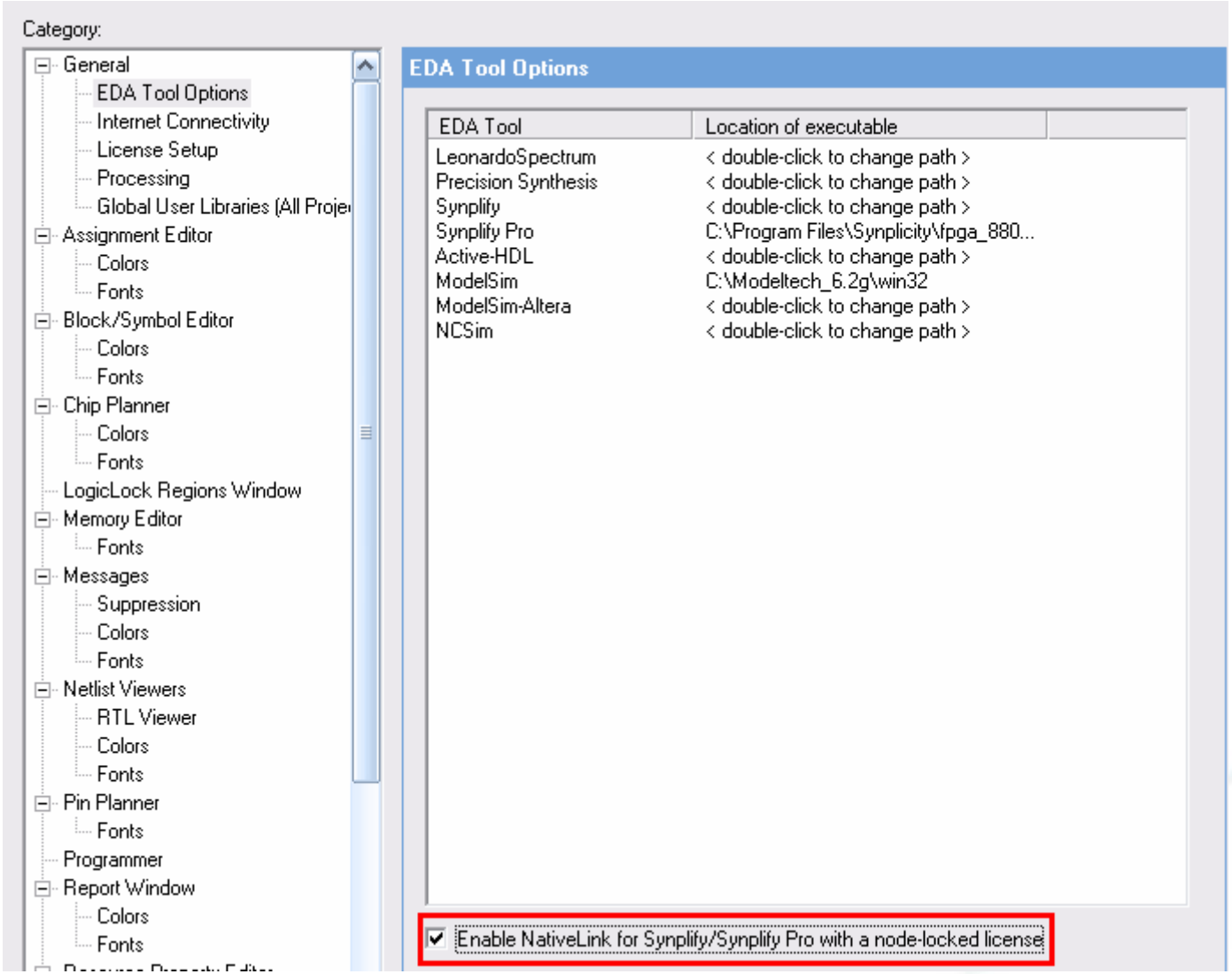
Cytech Technology

与Synthesis相关的设置(2)



- 优化目标：速度，面积和平衡，默认设置为平衡
- 一般是优化工程设计的第一步

第三方综合器Synplify Pro嵌入(1)

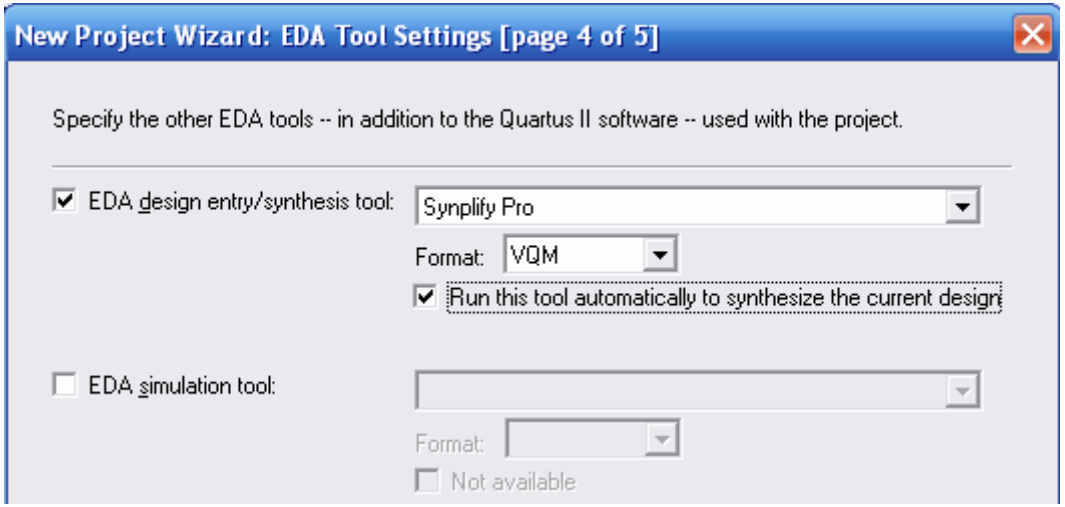


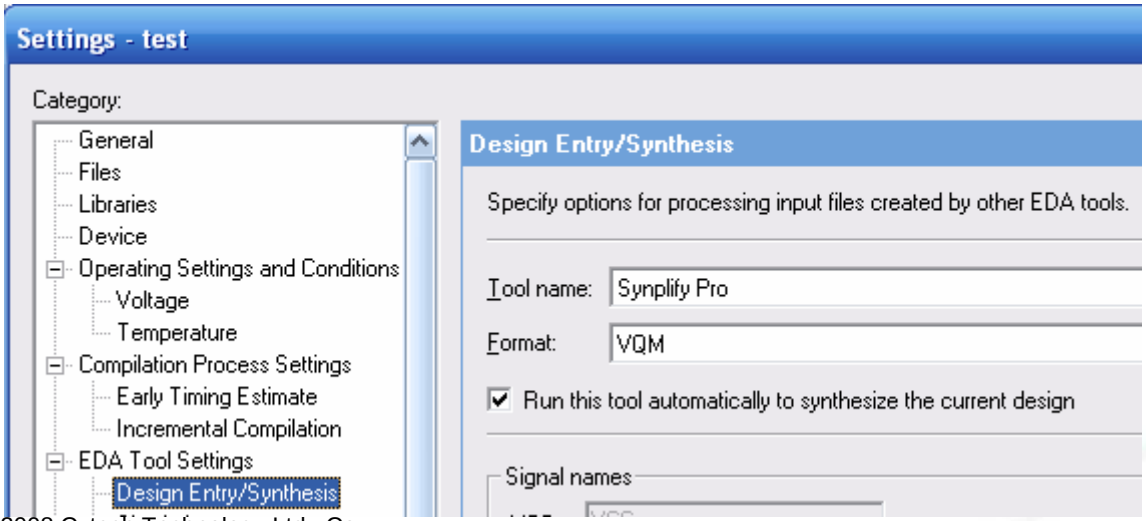
Step1:

Tools菜单Options->General->EDA Tool Options指定SynplifyPro的安装路径，选中”Enable NativeLink for Synplify/Synplify Pro with a node-locked license“ 。如果不选，QII是无法直接调用Synplify进行综合的。

Cytech Technology

第三方综合器Synplify Pro嵌入(2)





Step2:

-如果是新建工程，需要在EDA Tool Settings中设置，并选中“Run this tool antomatically to synthesize the current design”

-如果是现有工程，Settins->EDA Tool Settings同样设置

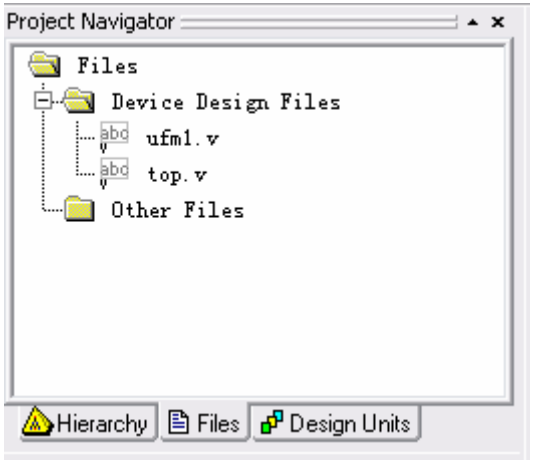
Cytech Technology

第三方综合器Synplify Pro嵌入(3)

-如果使用IPcore，QII导航界面可以看到的是IPcore生成的.v文件，这样是无法进行直接调用SynplifyPro的，软件会报错

-只需要将ufm1.v文件删除，用ufm1_bb.v文件替代，就可以直接调用Synplify Pro

-QII会自动在工程目录下生成synplify_xxx_work目录，目录下有供Synplify Pro可以直接打开的工程文件



Cytech Technology

Quartus II 软件使用教程

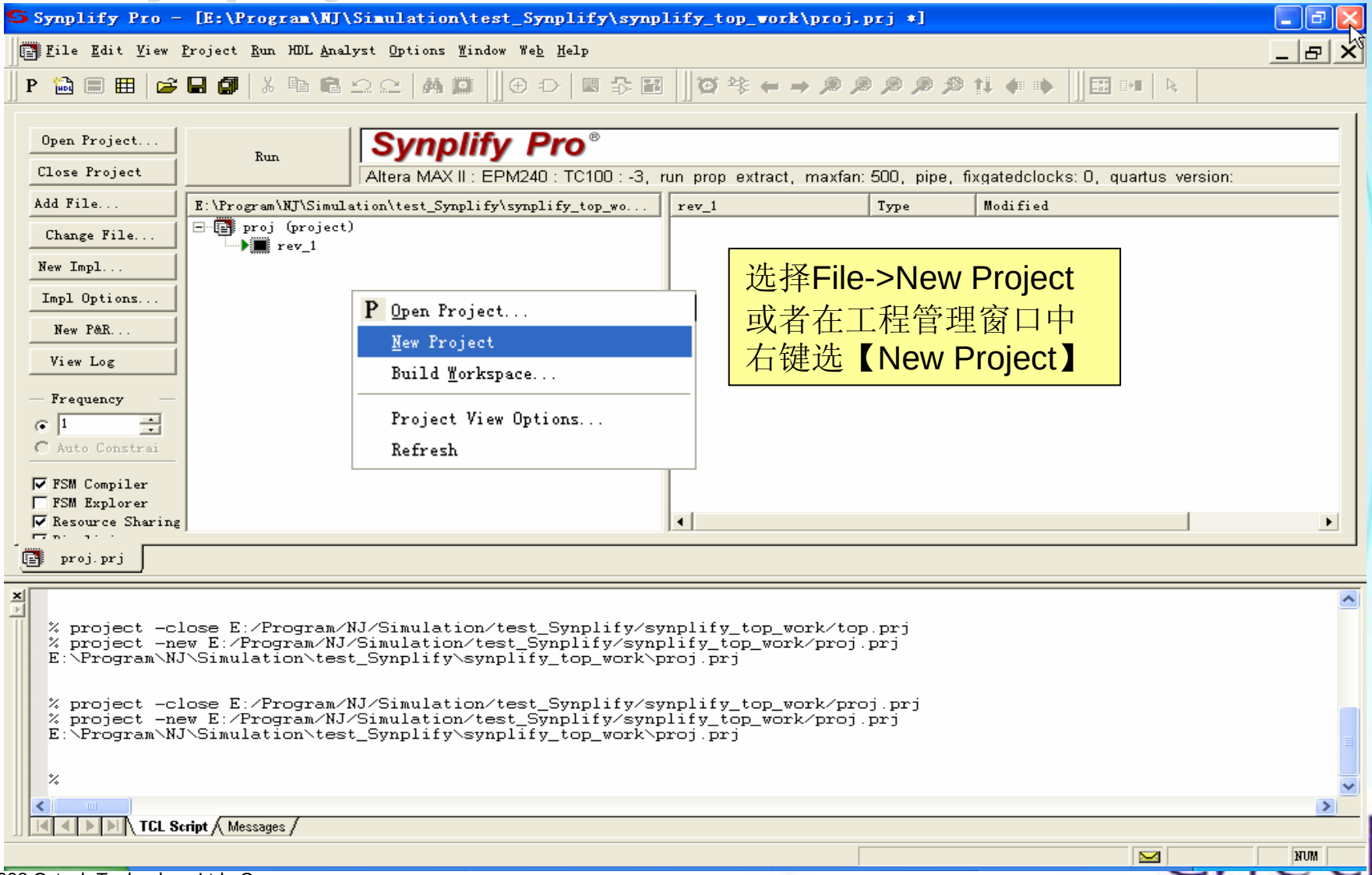
使用Synplify Pro做综合

使用Synplify Pro做综合

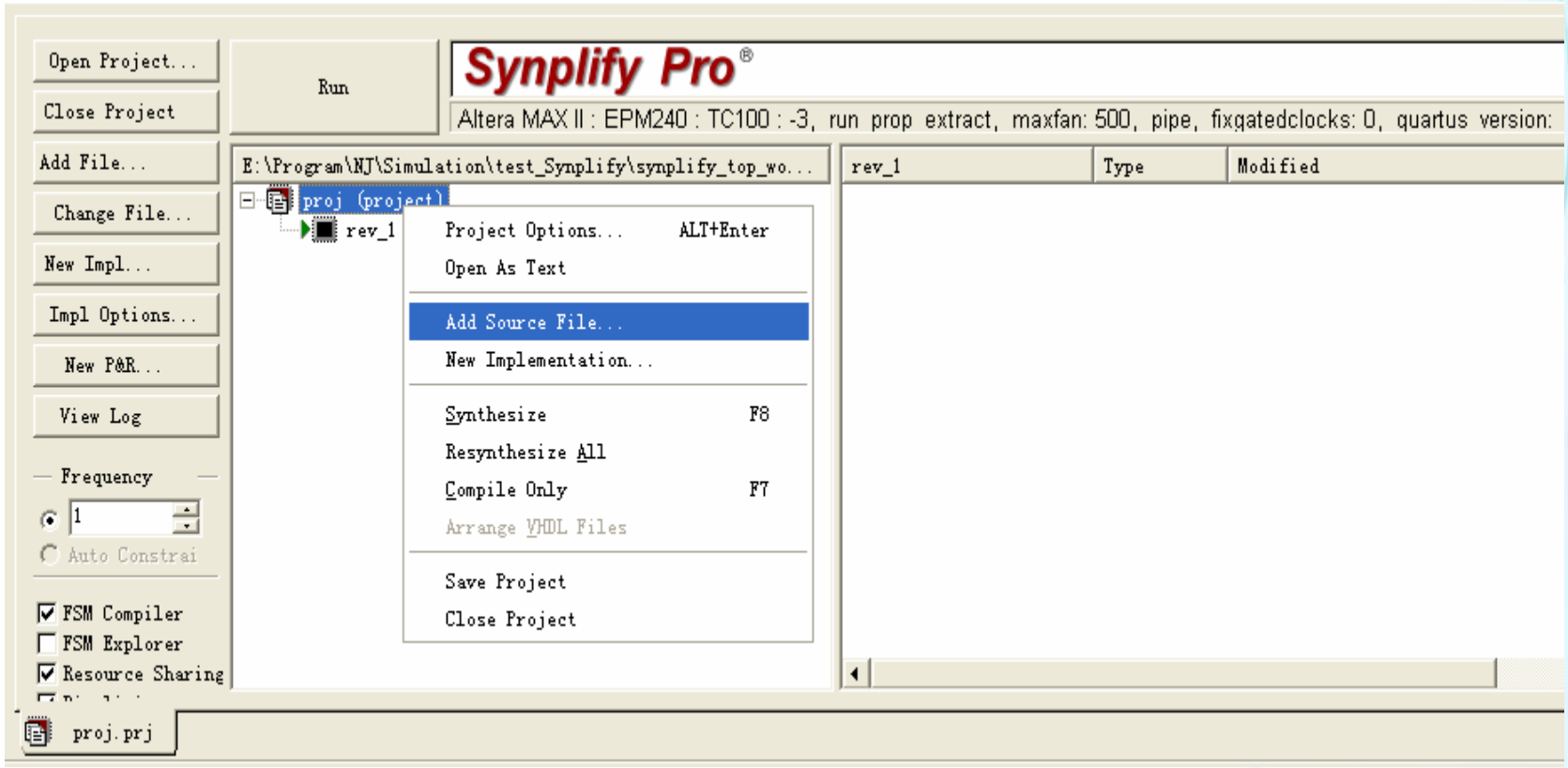
- 通常我们综合时，使用的是Quartus II自带的综合工具，实际工作中，许多设计人员都习惯于使用专业综合工具Synplify Pro。
- 正常情况下，正版的Quartus II和Synplify Pro可以实现无缝链接，过程同ISE差不多。但是大多数用户使用时，Quartus II直接调用Synplify Pro往往是有问题的，因此我们最好将两者分离开来操作。

Cytech Technology

在Synplify Pro下建立工程

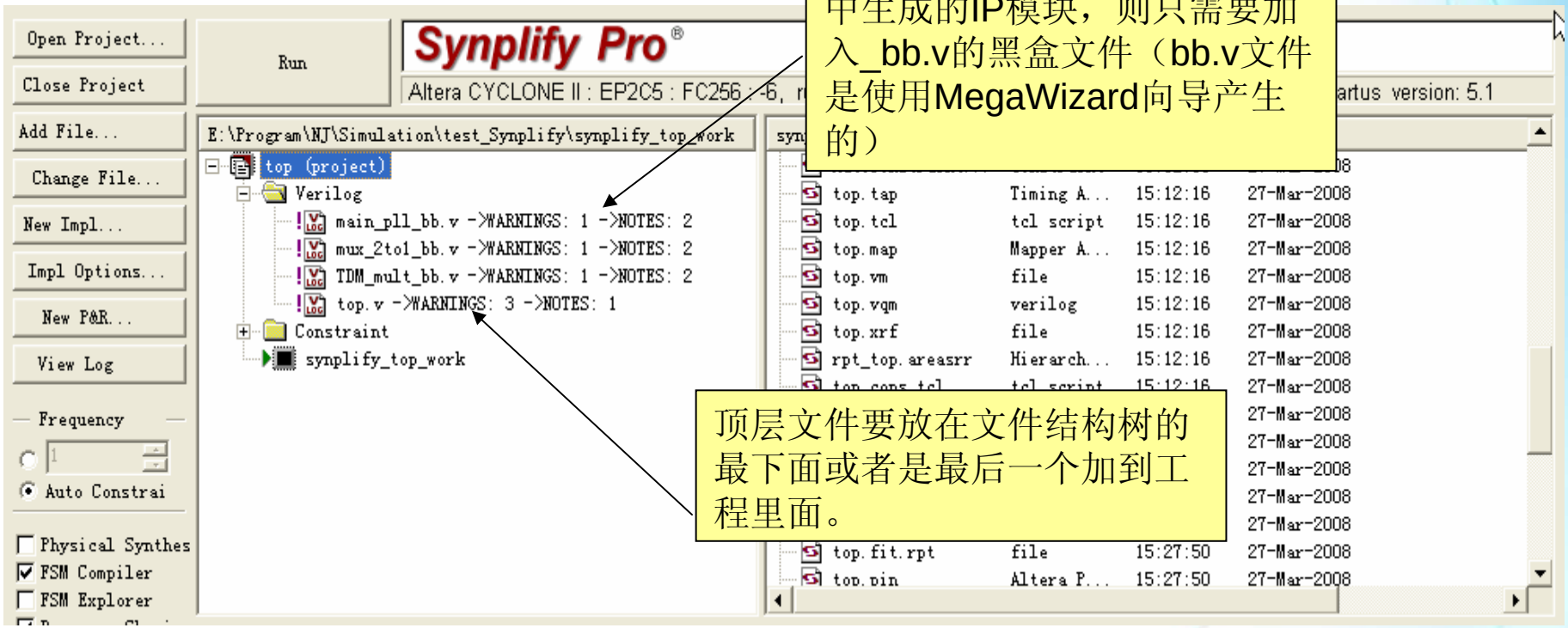


添加源文件



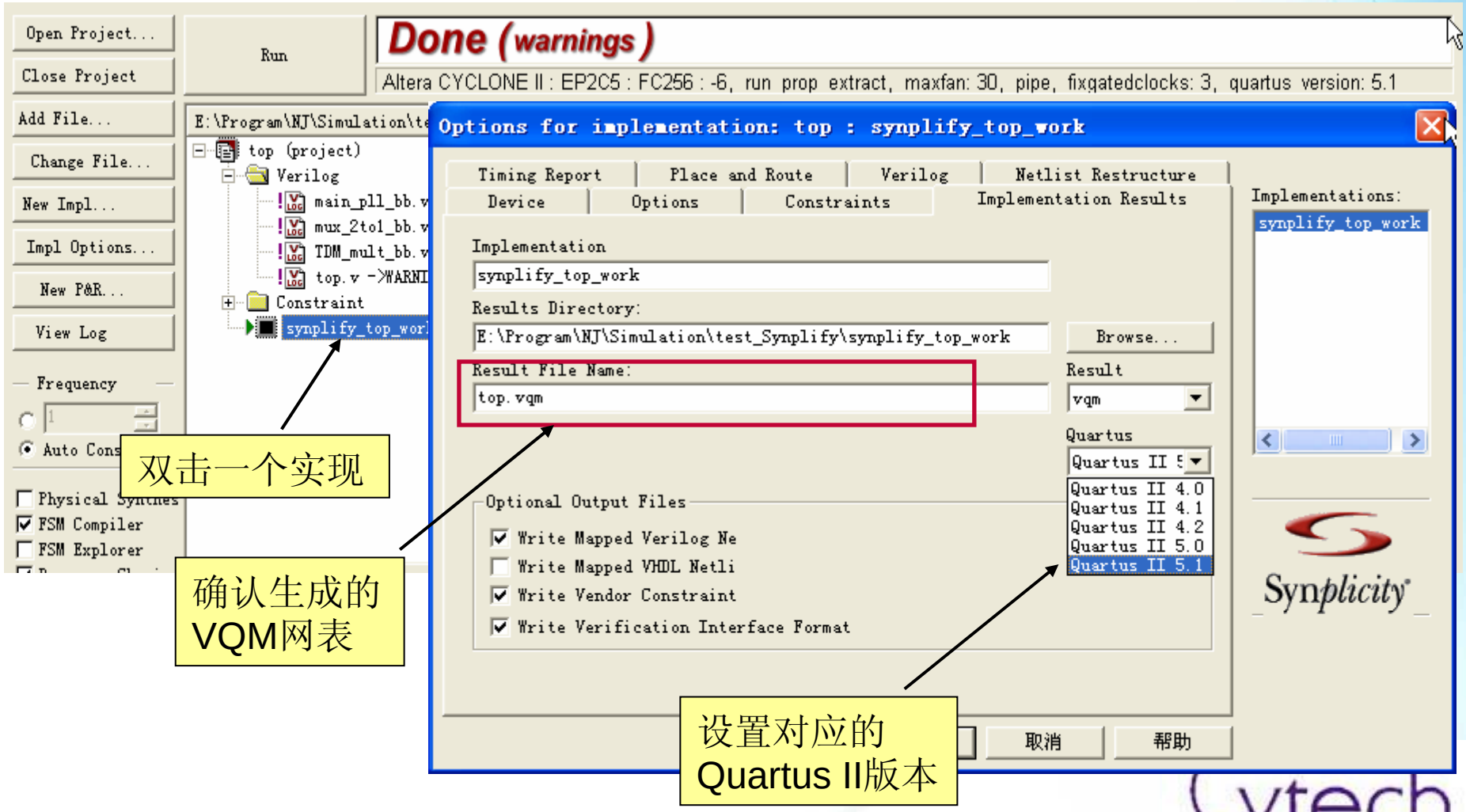
Cytech Technology

添加源文件时注意：



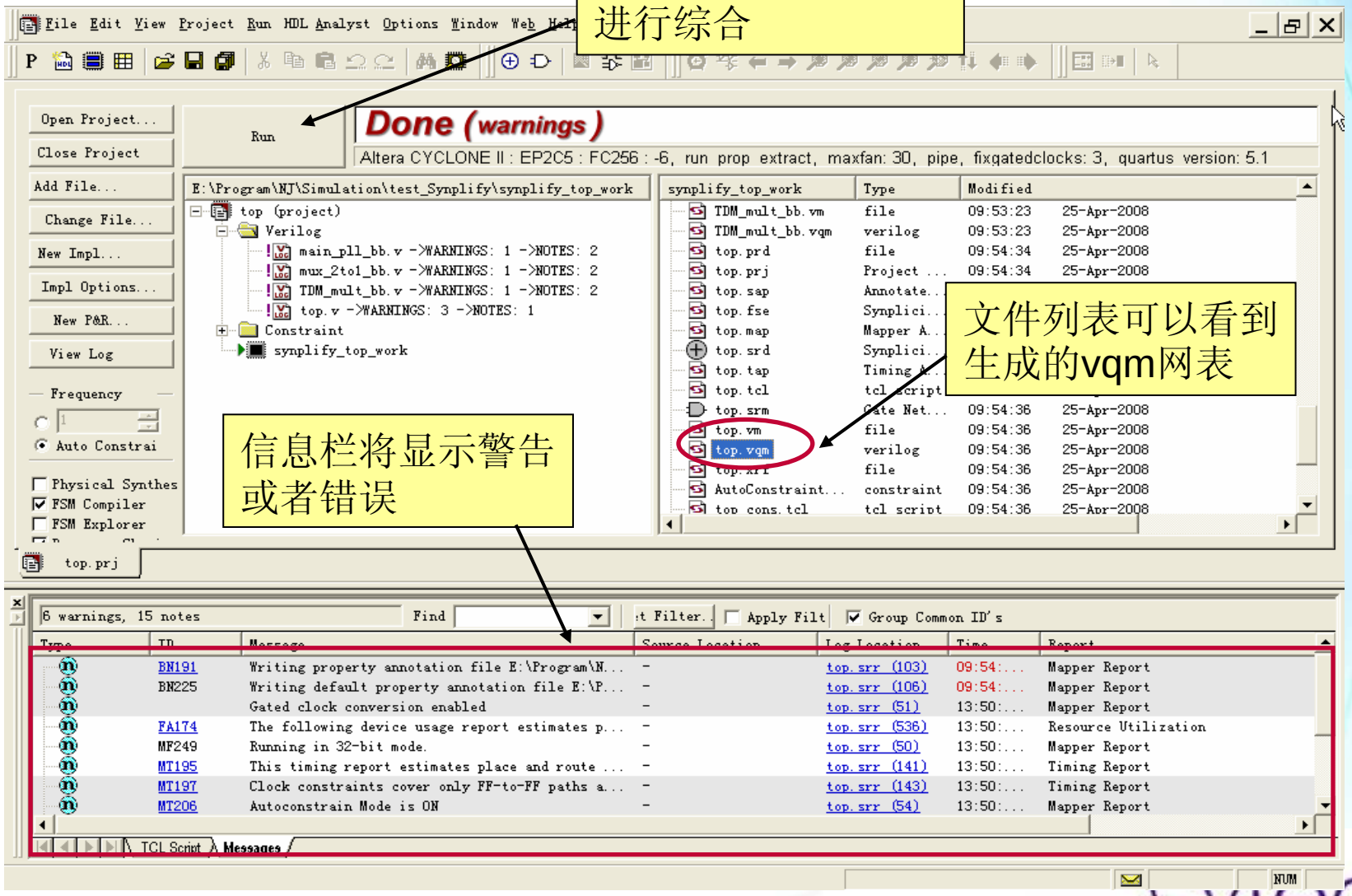
Cytech Technology

设置工程属性



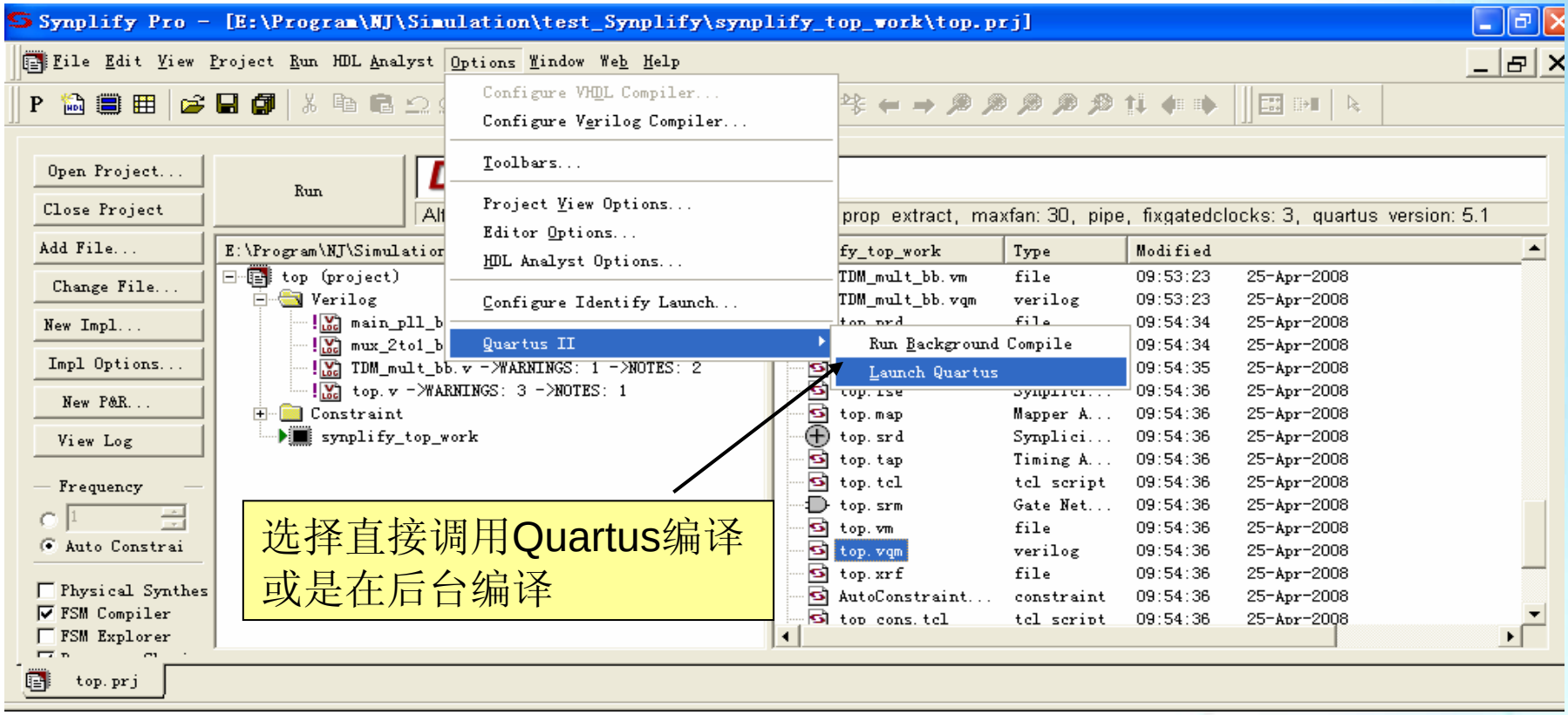
Cytech Technology

综合设计



Cytech Technology

调用Quartus II编译工程

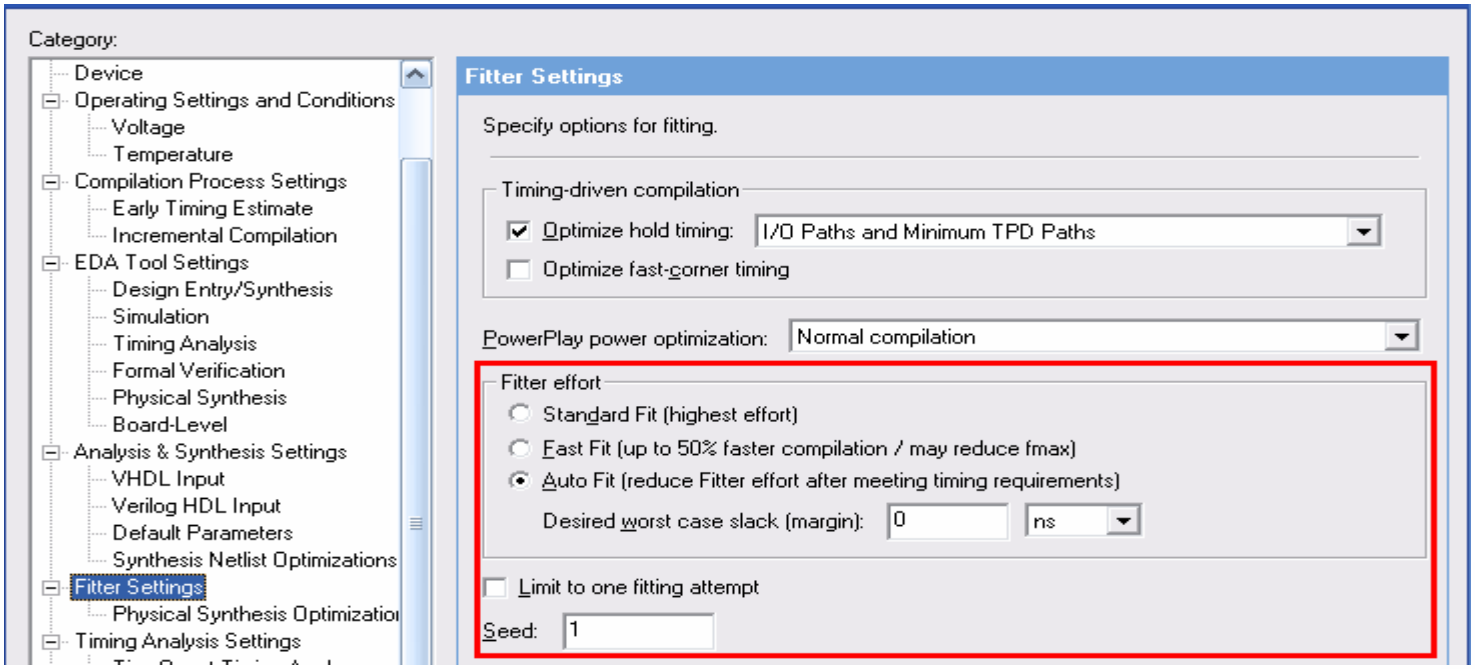


需要注意的是，在Synplify Pro工程下用到的IP模块调用的是_bb.v的文件，但进入到Quartus II 中则需要的是由MegaWizard向导生成的.v文件，所以最好让Quartus II 工程文件与IP生成的.v文件放在同一个目录下。

Quartus II 软件使用教程

布局布线

与Fitter相关的设置(1)



-Standard Fit

编译效果最好，时间最长

-Fast Fit

编译时间减少50%，牺牲Fmax作代价

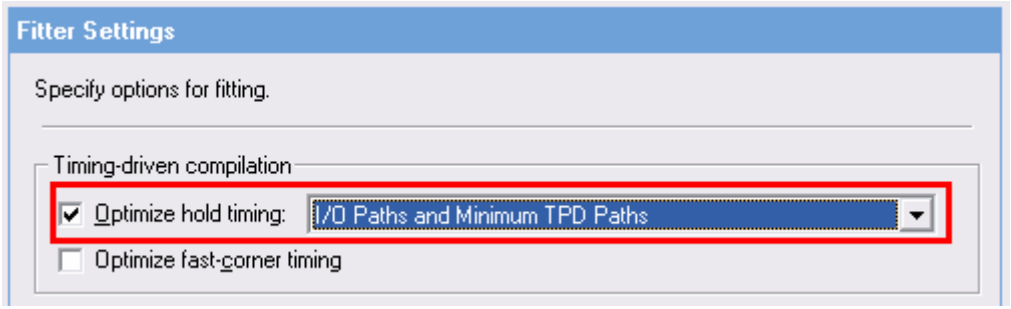
-Auto Fit

一旦满足时序要求，QII会停止优化，从而节省编译时间

-One fitting attempt（不同的种子导致编译结果小幅度变动，波动范围在±**5%**）

Cytech Technology

与Fitter相关的设置(2)



- 默认情况下为 **“IO to register and min Tco”**
  - -表示以IO到寄存器的Th约束、从寄存器到IO的最小Tco约束和从IO或寄存器到IO或寄存器的最小TPD约束为优化目标

- 设置为 **“All Paths”**
  - -除了IO路径和最小TPD路径为优化目标外，增加了寄存器到寄存器的时序约束优化
  - -尽可能的使用同步设计以避免时序问题

- 对于时序报告中**Hold** 时间的不满足，可以尝试设置**“All Paths”**

Cytech Technology

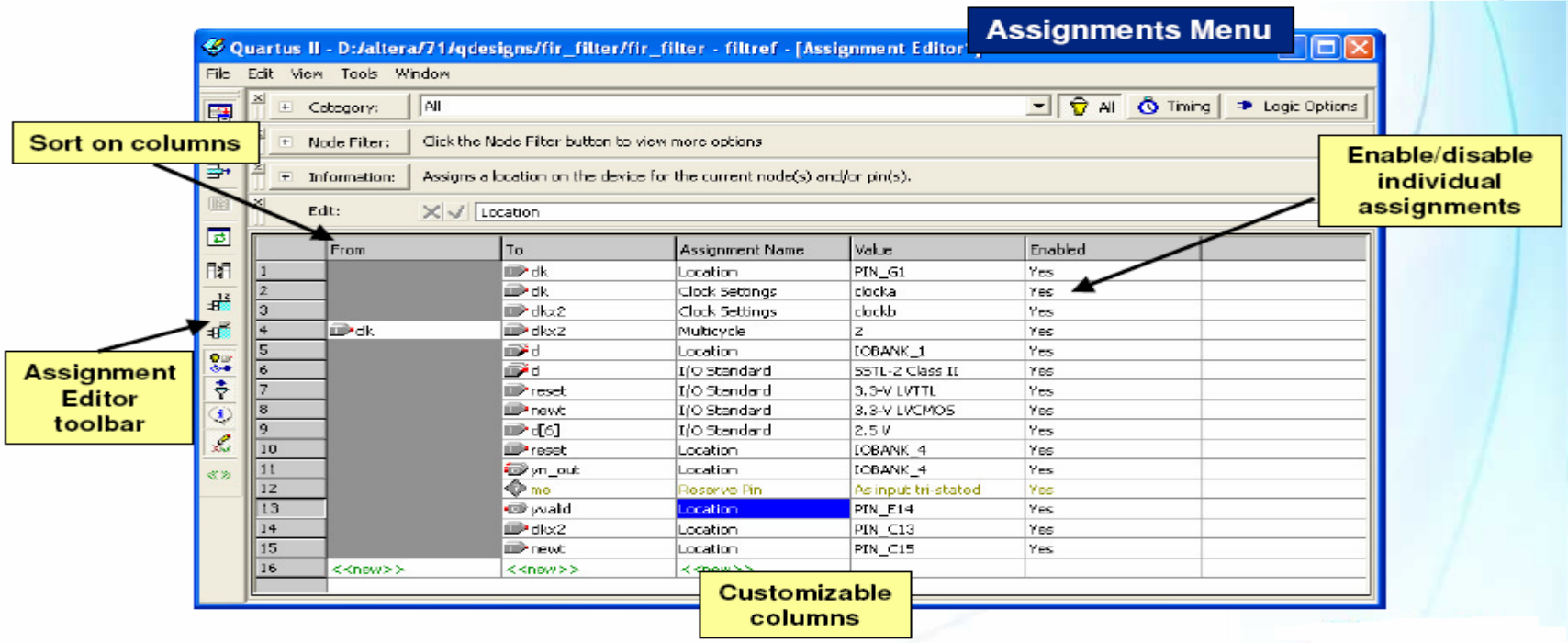
Quartus II 软件使用教程

Assignment Editor

Assignment Editor (AE)

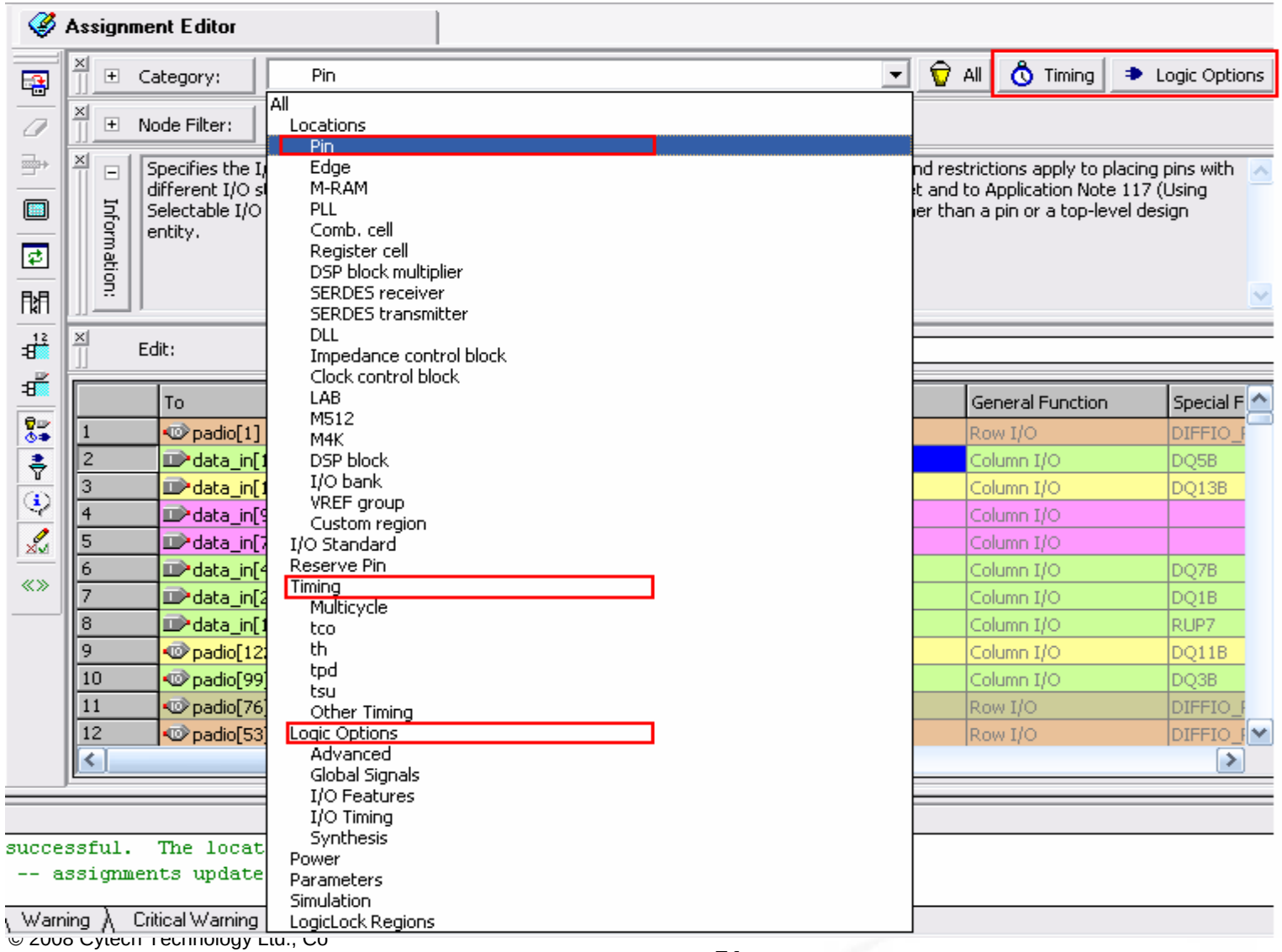
- 选择菜单Assignments→Assigment Editor或者直接点击按钮

- 通过AE生成的各种约束都会保存在.QSF文件中



Cytech Technology

Using AE

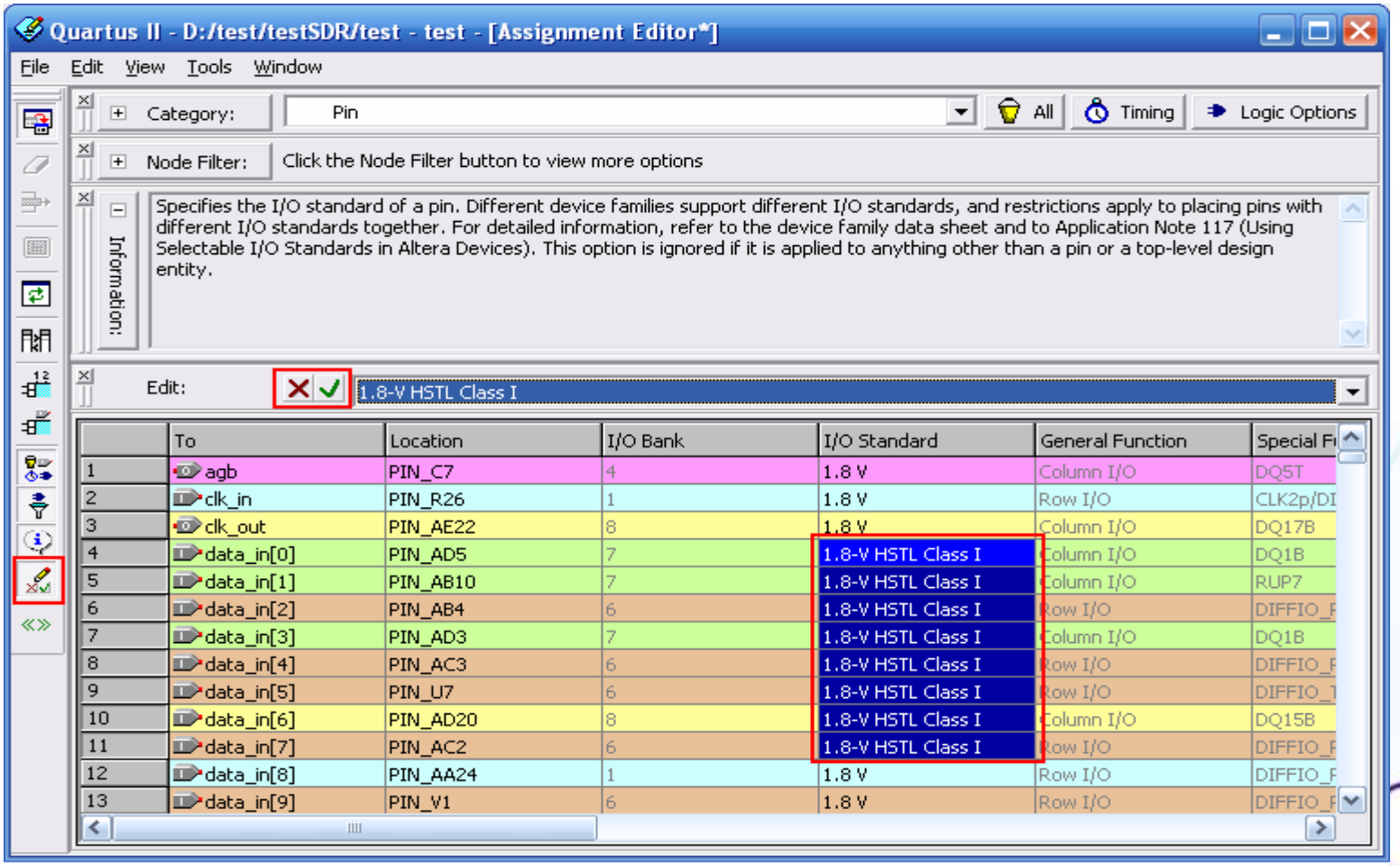


Category下拉选择，一般常用的是Pin, Timing和Logic Options，可以分别独立设置

Cytech Technology

编辑多个约束

- Use Edit bar, auto-fill, copy & paste



	To	Location	I/O Bank	I/O Standard	General Function	Special F
1	agb	PIN_C7	4	1.8 V	Column I/O	DQ5T
2	clk_in	PIN_R26	1	1.8 V	Row I/O	CLK2p/DI
3	clk_out	PIN_AE22	8	1.8 V	Column I/O	DQ17B
4	data_in[0]	PIN_AD5	7	1.8-V HSTL Class I	Column I/O	DQ1B
5	data_in[1]	PIN_AB10	7	1.8-V HSTL Class I	Column I/O	RUP7
6	data_in[2]	PIN_AB4	6	1.8-V HSTL Class I	Row I/O	DIFFIO_
7	data_in[3]	PIN_AD3	7	1.8-V HSTL Class I	Column I/O	DQ1B
8	data_in[4]	PIN_AC3	6	1.8-V HSTL Class I	Row I/O	DIFFIO_
9	data_in[5]	PIN_U7	6	1.8-V HSTL Class I	Row I/O	DIFFIO_
10	data_in[6]	PIN_AD20	8	1.8-V HSTL Class I	Column I/O	DQ15B
11	data_in[7]	PIN_AC2	6	1.8-V HSTL Class I	Row I/O	DIFFIO_
12	data_in[8]	PIN_AA24	1	1.8 V	Row I/O	DIFFIO_
13	data_in[9]	PIN_V1	6	1.8 V	Row I/O	DIFFIO_

鼠标选择多个需改动的约束，在Edit Bar中下拉或直接复制粘贴选择新的约束。当Edit Bar中的“√”变成灰色表示修改成功

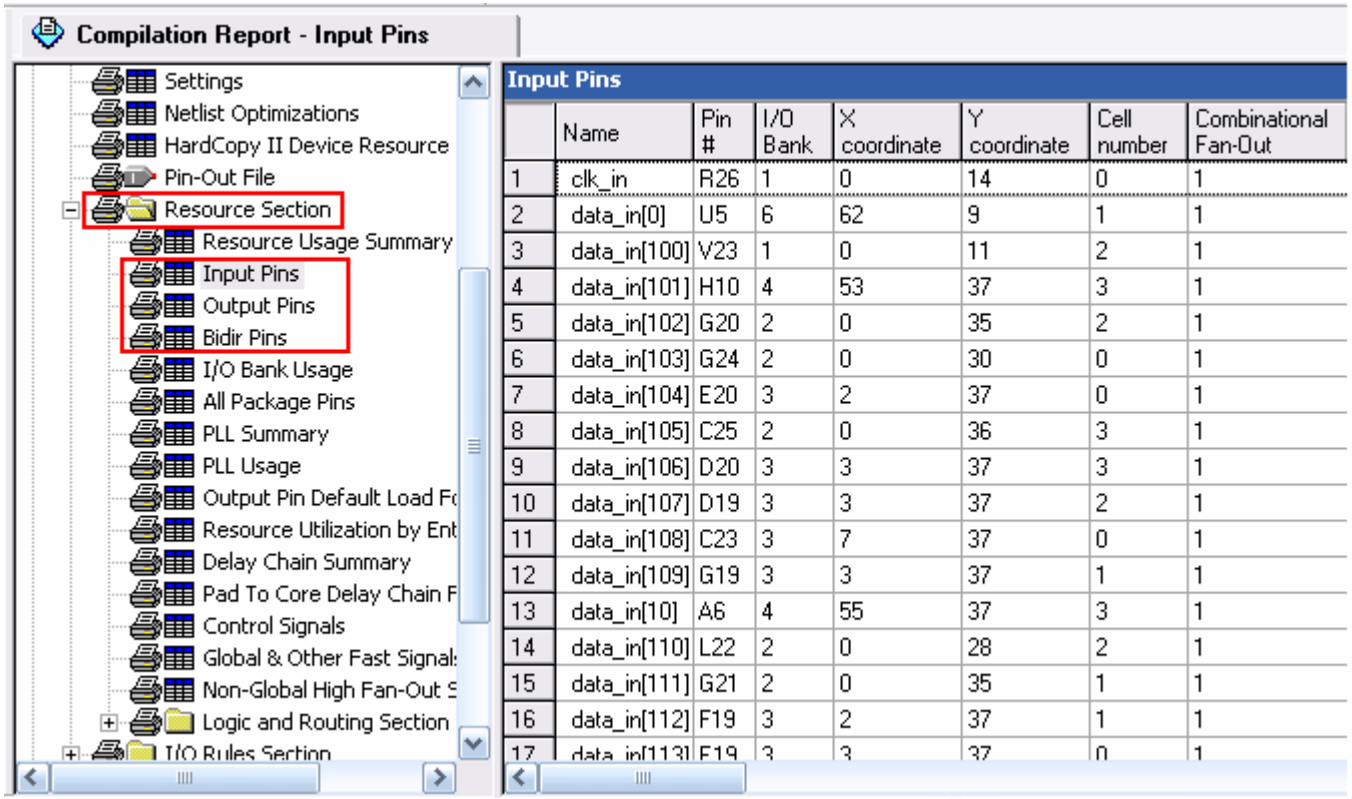
Cytech Technology

Quartus II 软件使用教程

管脚分配

Pin的约束(1)

- 一个新建Project是没有约束的，AE显示为空
- 如果在没有约束的情况下直接编译Project，QII会自动分配引脚，显示在编译报告中(Compilation Report->Fitter->Resource Section->Input Pins, Output Pins or Bidir Pins)



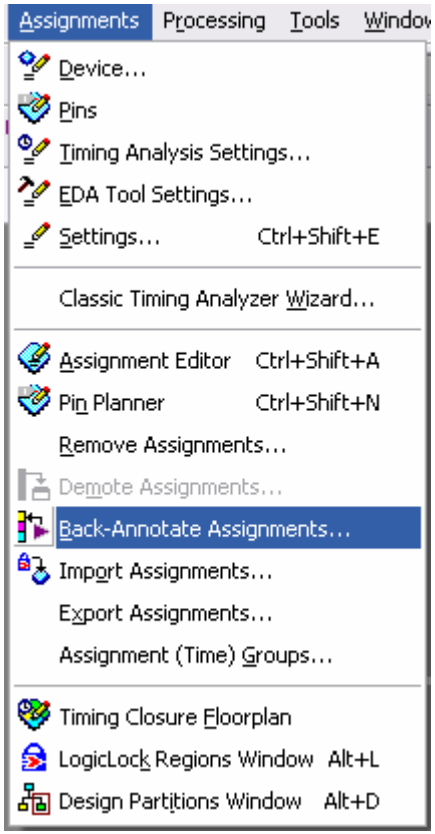
Input Pins

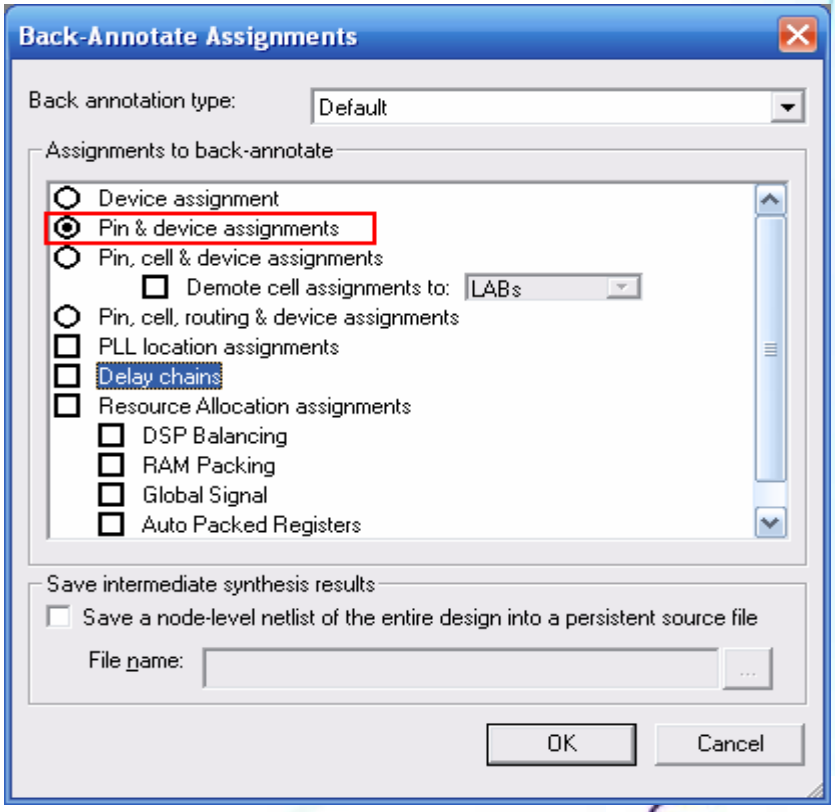
	Name	Pin #	I/O Bank	X coordinate	Y coordinate	Cell number	Combinational Fan-Out
1	clk_in	R26	1	0	14	0	1
2	data_in[0]	U5	6	62	9	1	1
3	data_in[100]	V23	1	0	11	2	1
4	data_in[101]	H10	4	53	37	3	1
5	data_in[102]	G20	2	0	35	2	1
6	data_in[103]	G24	2	0	30	0	1
7	data_in[104]	E20	3	2	37	0	1
8	data_in[105]	C25	2	0	36	3	1
9	data_in[106]	D20	3	3	37	3	1
10	data_in[107]	D19	3	3	37	2	1
11	data_in[108]	C23	3	7	37	0	1
12	data_in[109]	G19	3	3	37	1	1
13	data_in[10]	A6	4	55	37	3	1
14	data_in[110]	L22	2	0	28	2	1
15	data_in[111]	G21	2	0	35	1	1
16	data_in[112]	F19	3	2	37	1	1
17	data_in[113]	E19	3	3	37	0	1

Cytech Technology

Pin的约束(2)

- 如果需要手工输入Pin约束，除直接编辑QSF文件之外，还有两种相对来说比较方便的方法
- 方法一：反标约束，让QII自动生成约束

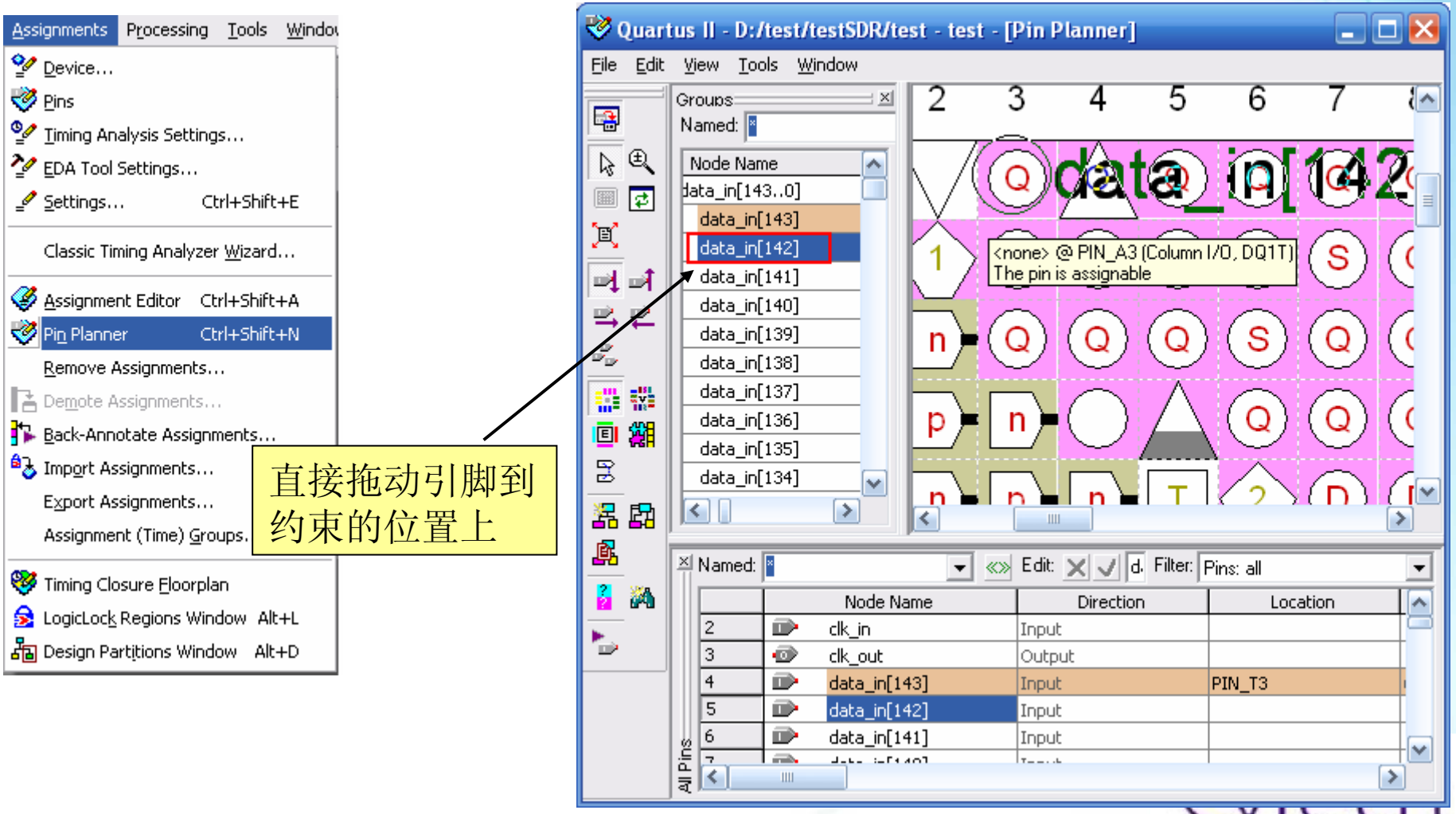




反标时只选择“Pin & device assignments”，AE立刻显示反标过的Pin约束

Pin的约束(3)

- 方法二： Pin Planner图形化操作

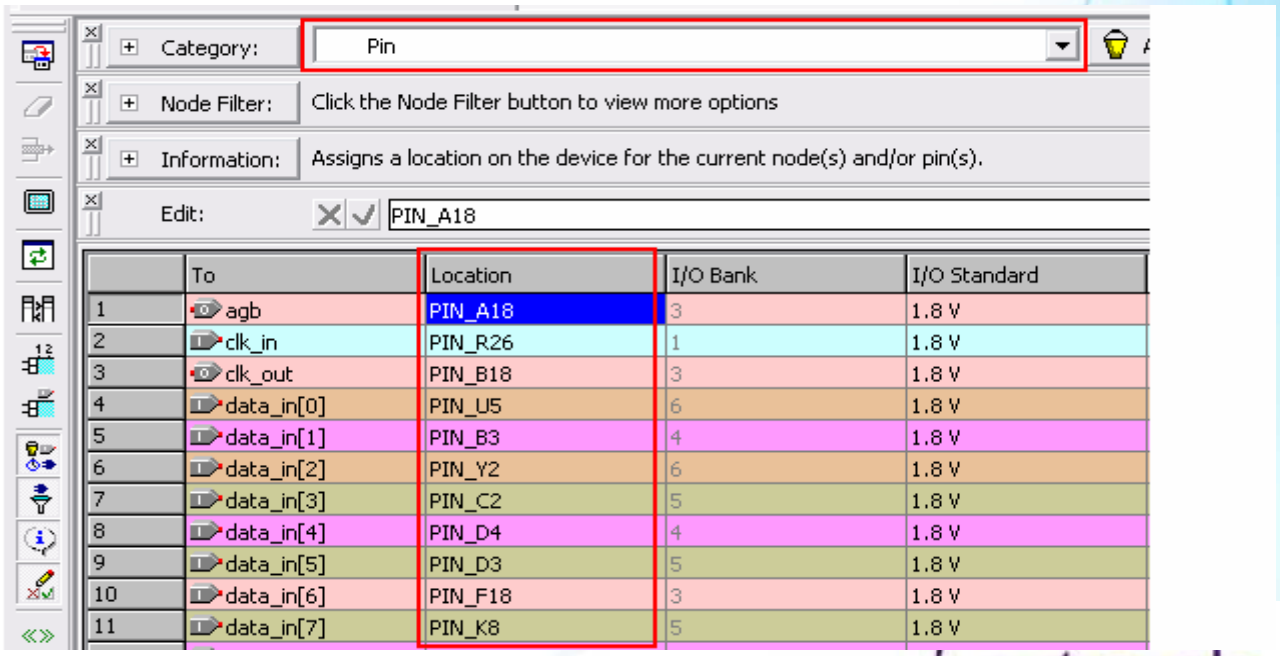


Pin的约束(4)

- 需要注意的是，在反标FPGA时会多出两三个信号，这与FPGA的配置模式有关。为避免麻烦，建议直接删除

	From	To	Assignment Name
292		reset	Location
293			Partition Hierarchy
294		~ASDO~	Location
295		~DATA0~	Location
296		~nCSO~	Location
297	<<new>>	<<new>>	<<new>>

在Category下拉选择Pin，直接修改Location栏下的Pin值就可以了。比如新约束为A22，点击“PIN_A18”，手工敲入“A22”即可

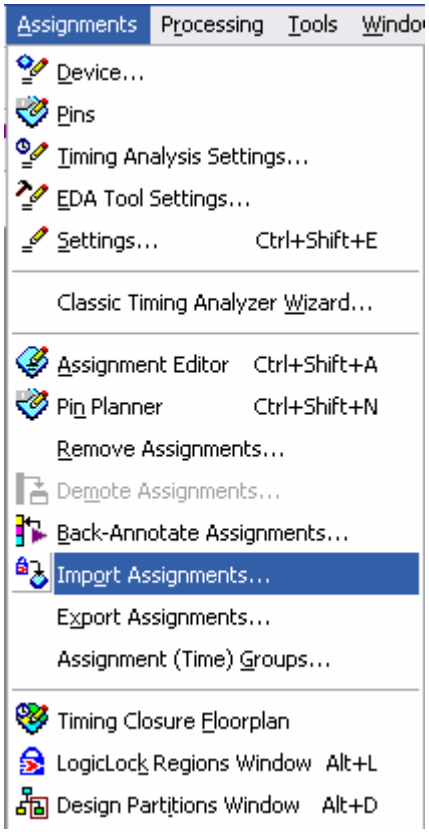


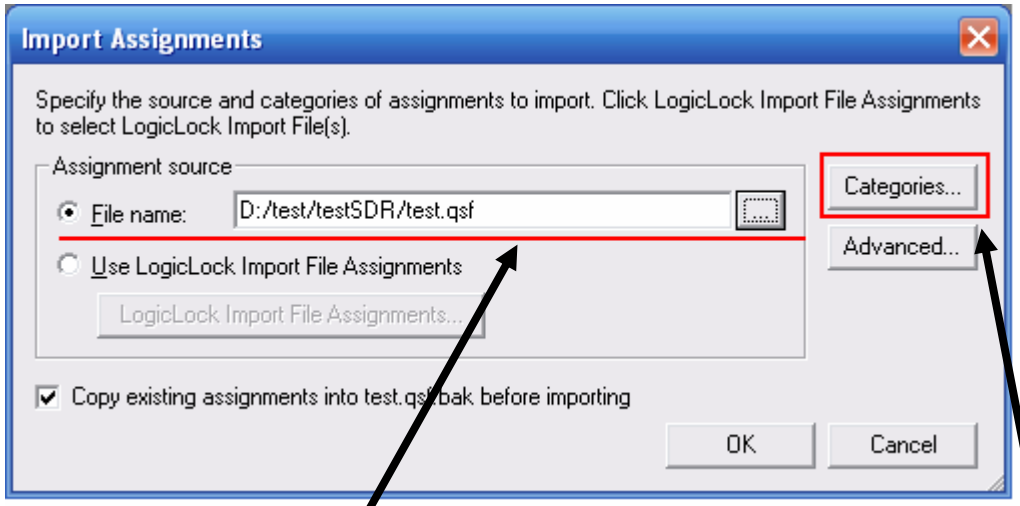
	To	Location	I/O Bank	I/O Standard
1	agb	PIN_A18	3	1.8 V
2	clk_in	PIN_R26	1	1.8 V
3	clk_out	PIN_B18	3	1.8 V
4	data_in[0]	PIN_U5	6	1.8 V
5	data_in[1]	PIN_B3	4	1.8 V
6	data_in[2]	PIN_Y2	6	1.8 V
7	data_in[3]	PIN_C2	5	1.8 V
8	data_in[4]	PIN_D4	4	1.8 V
9	data_in[5]	PIN_D3	5	1.8 V
10	data_in[6]	PIN_F18	3	1.8 V
11	data_in[7]	PIN_K8	5	1.8 V

Cytech Technology

Pin的约束(5)

- 如果先前已有.QSF文件，可以直接导入新的Project中，AE立刻显示新的约束





选择需导入的文件

选择需导入的约束类型

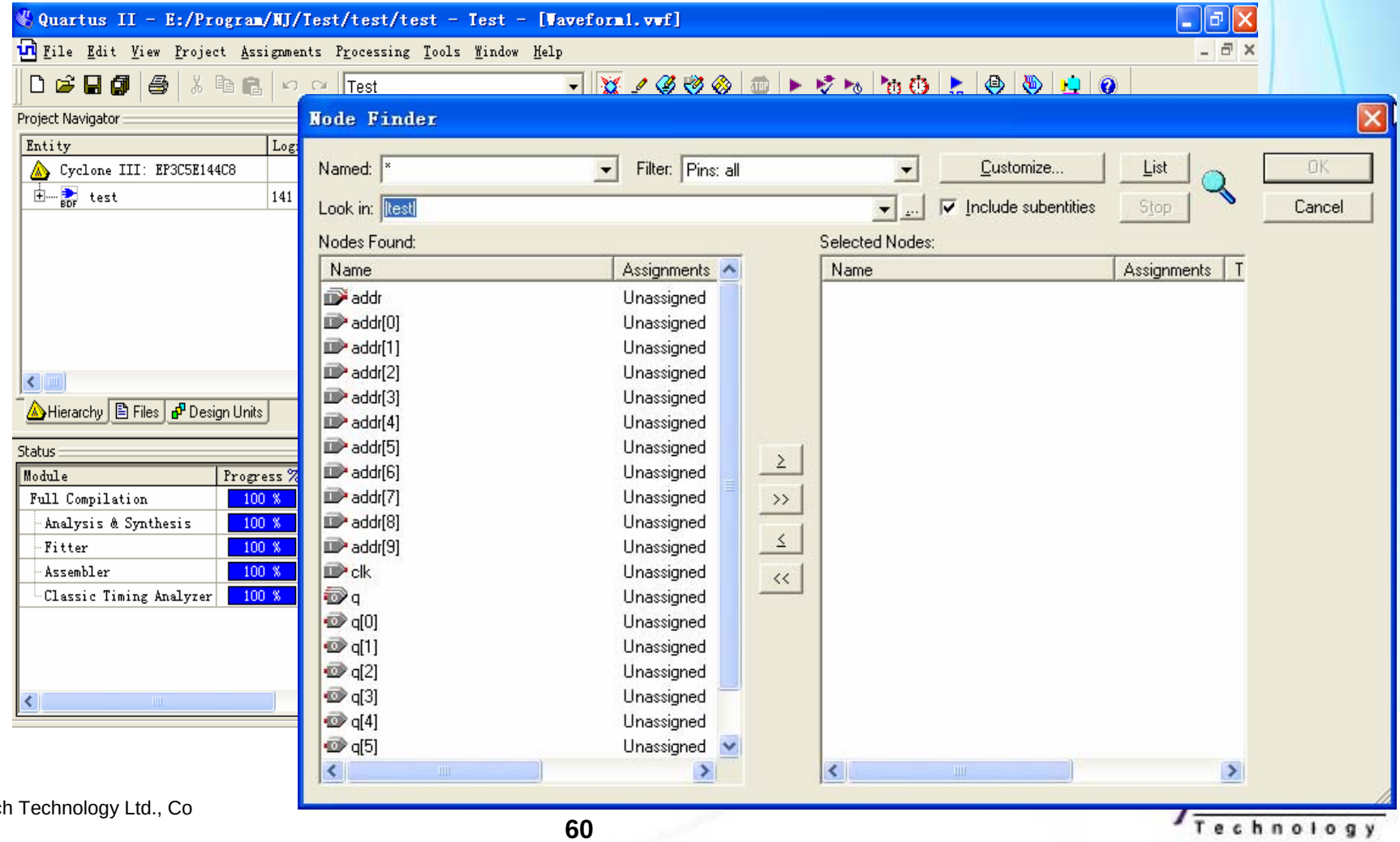
Cytech Technology

Quartus II 软件使用教程

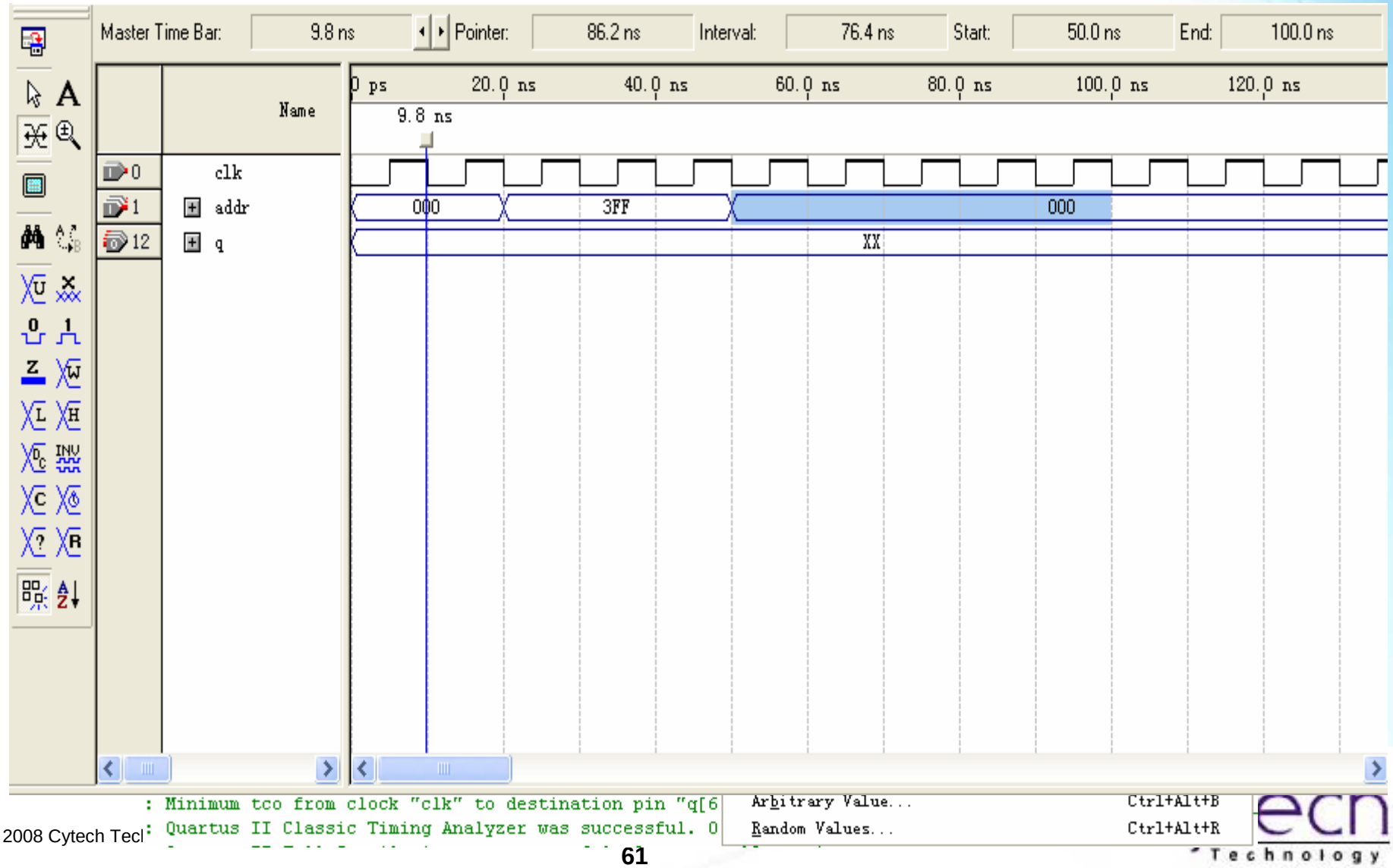
仿真

建立仿真文件

选择File菜单下的New->Other Files->Vector Waveform File



编辑波形

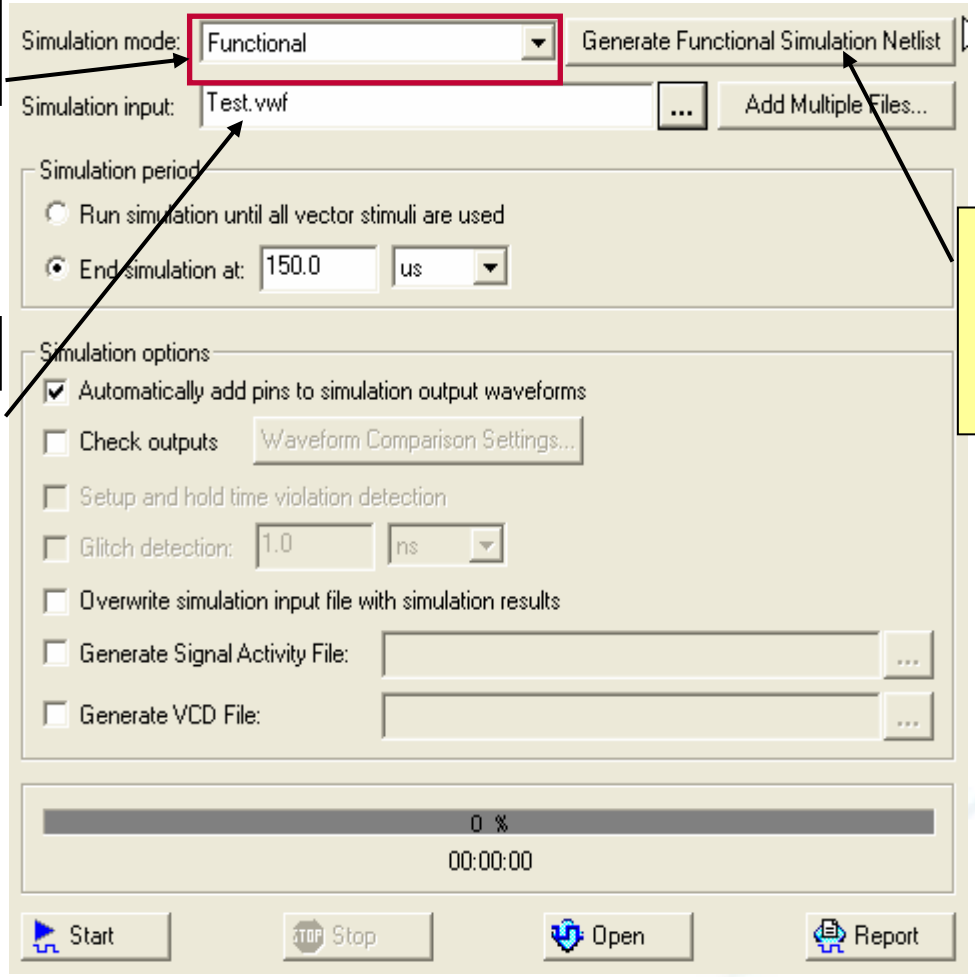


用Quartus II进行功能/时序仿真

- 从Processing菜单打开simulation tools

选择做功能仿真还是时序仿真

选择激励文件

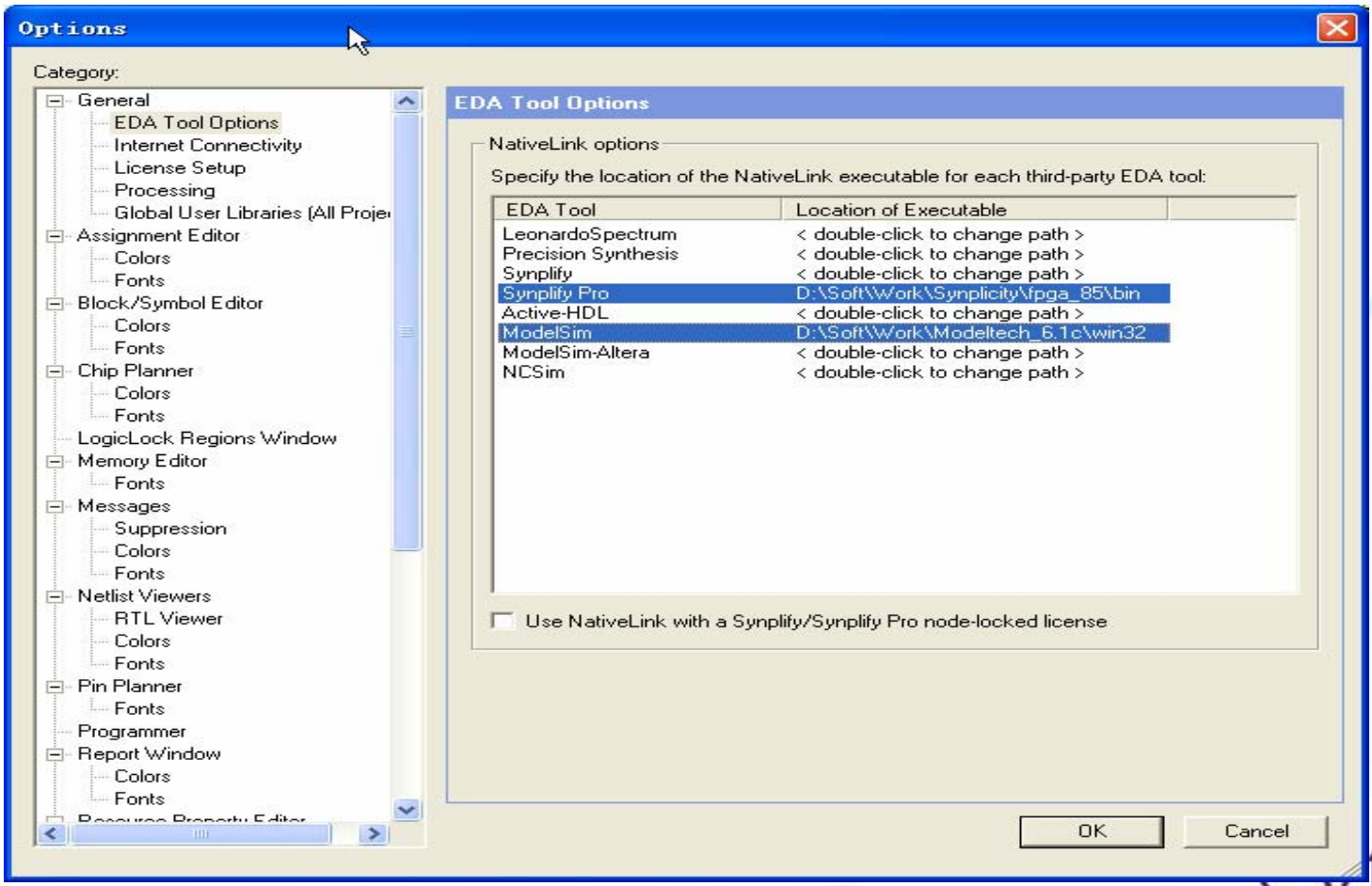


缺省情况下，QII产生的是timing netlist，因此在做功能仿真前，需要先生成功能仿真网表文件

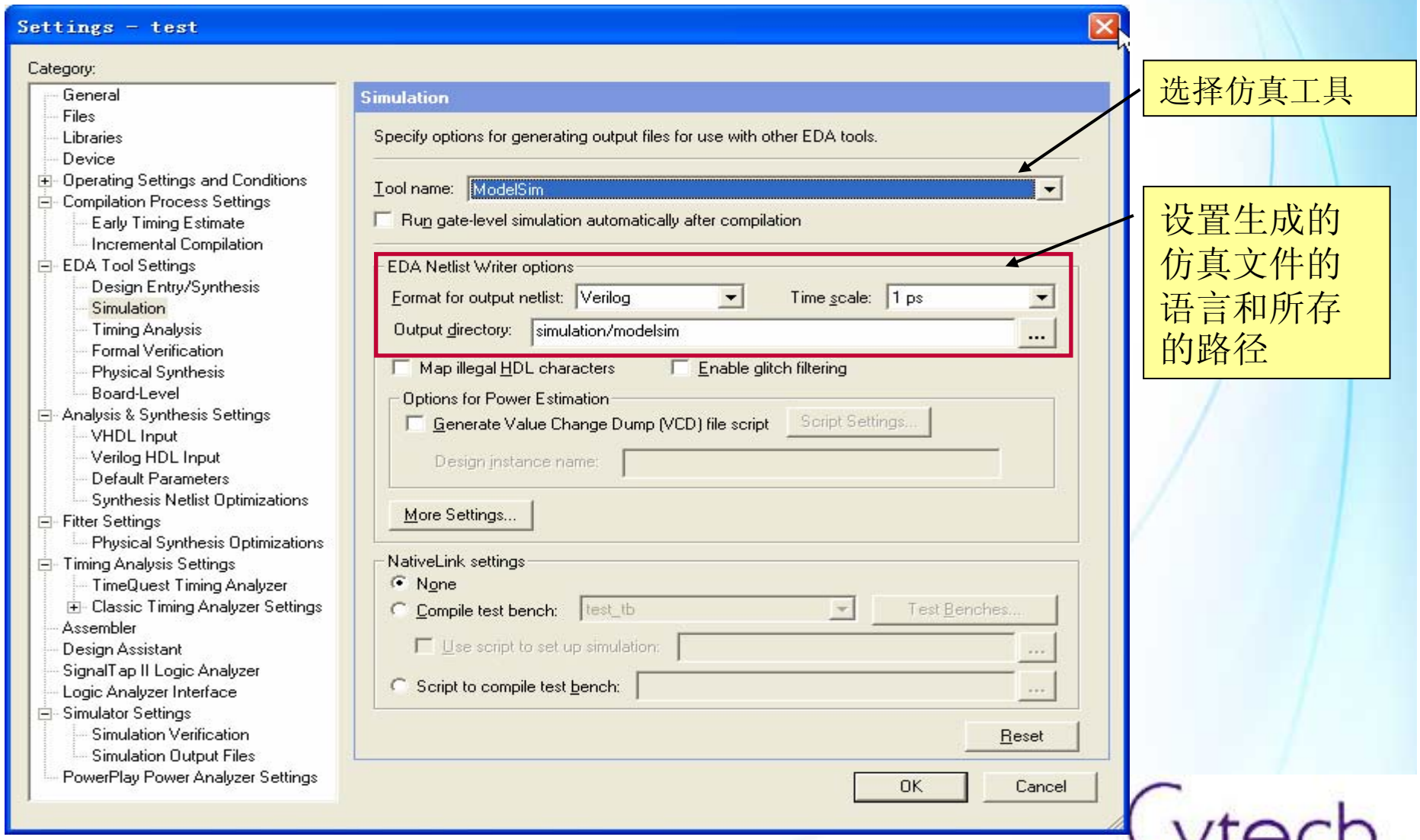
Cytech Technology

调用ModelSim-SE进行功能/时序仿真

1、选择Tools->Options，点击【General】/【EDA Tool Options】，设置ModelSim执行文件的安装路径（Synplify Pro也在此设置）。

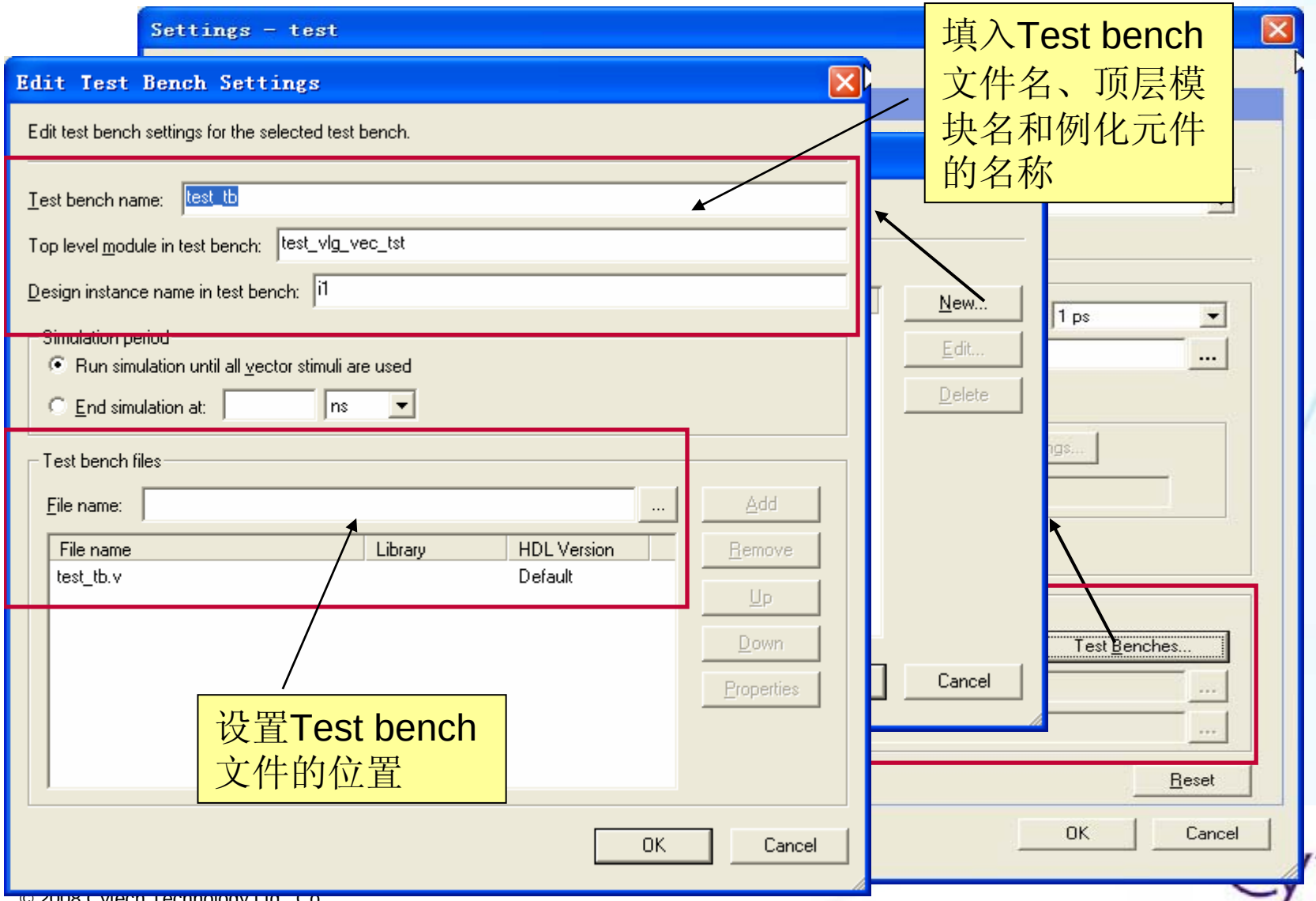


2、选择EDA Tools Settings下的Simulation栏，设置仿真工具 。



Cytech Technology

3、重点，设置TestBench！！！



Cytech Technology

4、运行仿真

- 点击  “Start Compilation”按钮编译工程，完成之后在当前的工程目录下可以看到一个名为“Simulation”的新文件夹，下面的“ModelSim”文件夹下包括仿真需要的.vo网表文件和包含延迟信息的.sdo文件。
- 如果之前在“Settings → EDA Tools Setting → Simulation”出现的设置栏中选中了“Run this tool automatically after compilation”，编译完成后Quartus会自动调用ModelSim进行门级时序仿真。
- 如果没选，则选择菜单Tools → EDA Simulation Tool → Run EDA Gate Level Simulation，Quartus便会调用ModelSim进行门级时序仿真。
- 选择Run EDA RTL Simulation则进行行为级仿真。
- 用户也可以单独在ModelSim中通过【Tools】/【Execute Macro】运行Quartus II自动生成的*_run_msim_gate_verilog.do文件进行时序仿真或者*_run_msim_rtl_verilog.do文件进行功能仿真 。

Cytech Technology

Quartus II 软件使用教程

器件编程

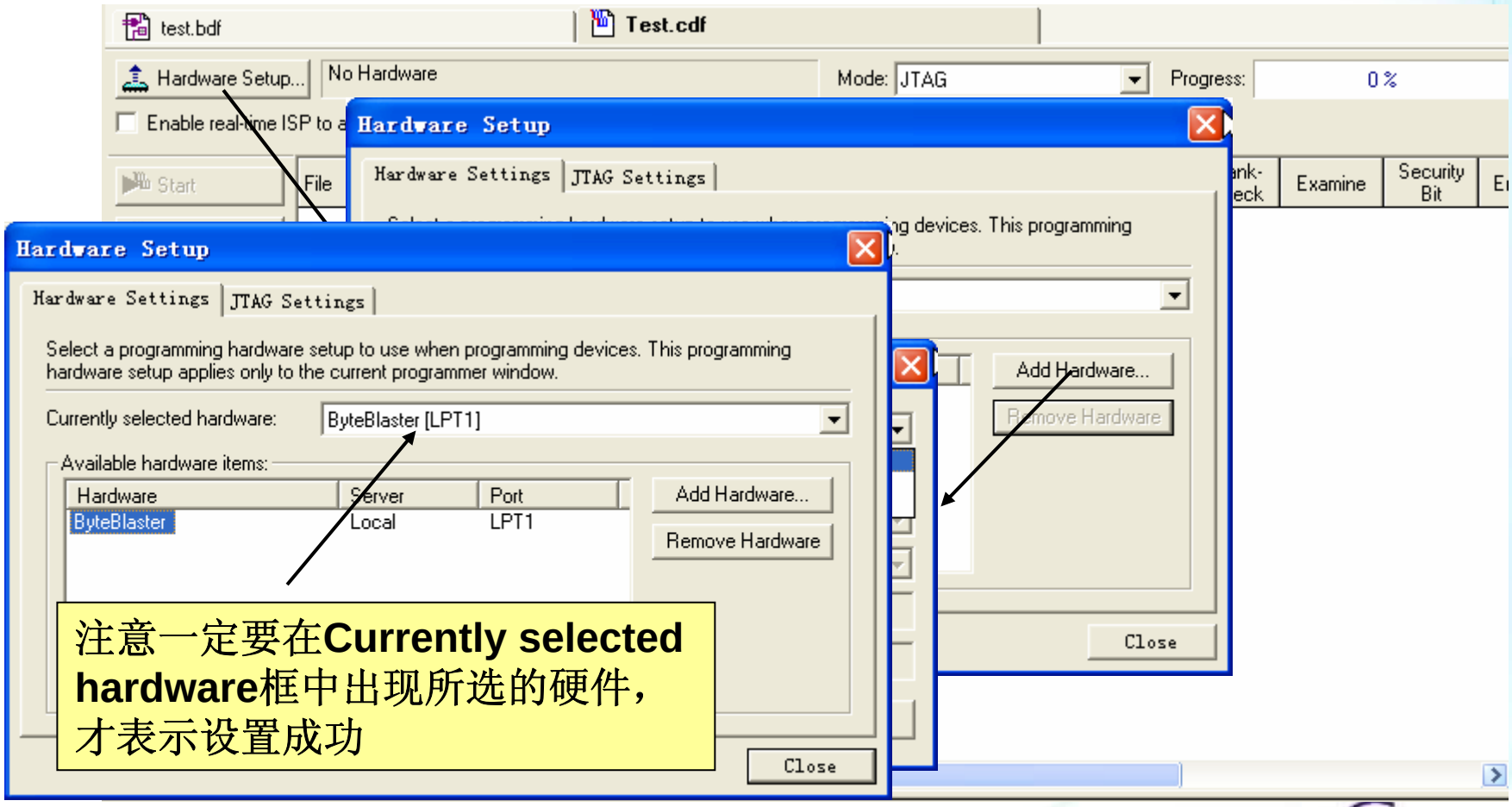
器件编程

- Alrera编程器硬件包括MasterBlaster™、ByteBlasterMV™、 ByteBlaster™ II、USB-Blaster和Ethernet Blaster下载电缆，或 Altera 编程单元 (APU)。
- Quartus II软件编程器具有四种编程模式：
  - 被动串行模式（Passive Serial mode）;
  - JTAG模式;
  - 主动串行编程模式（Active Serial Programming mode）;
  - 套接字内编程模式（In-Socket Programming mode）。

Cytech Technology

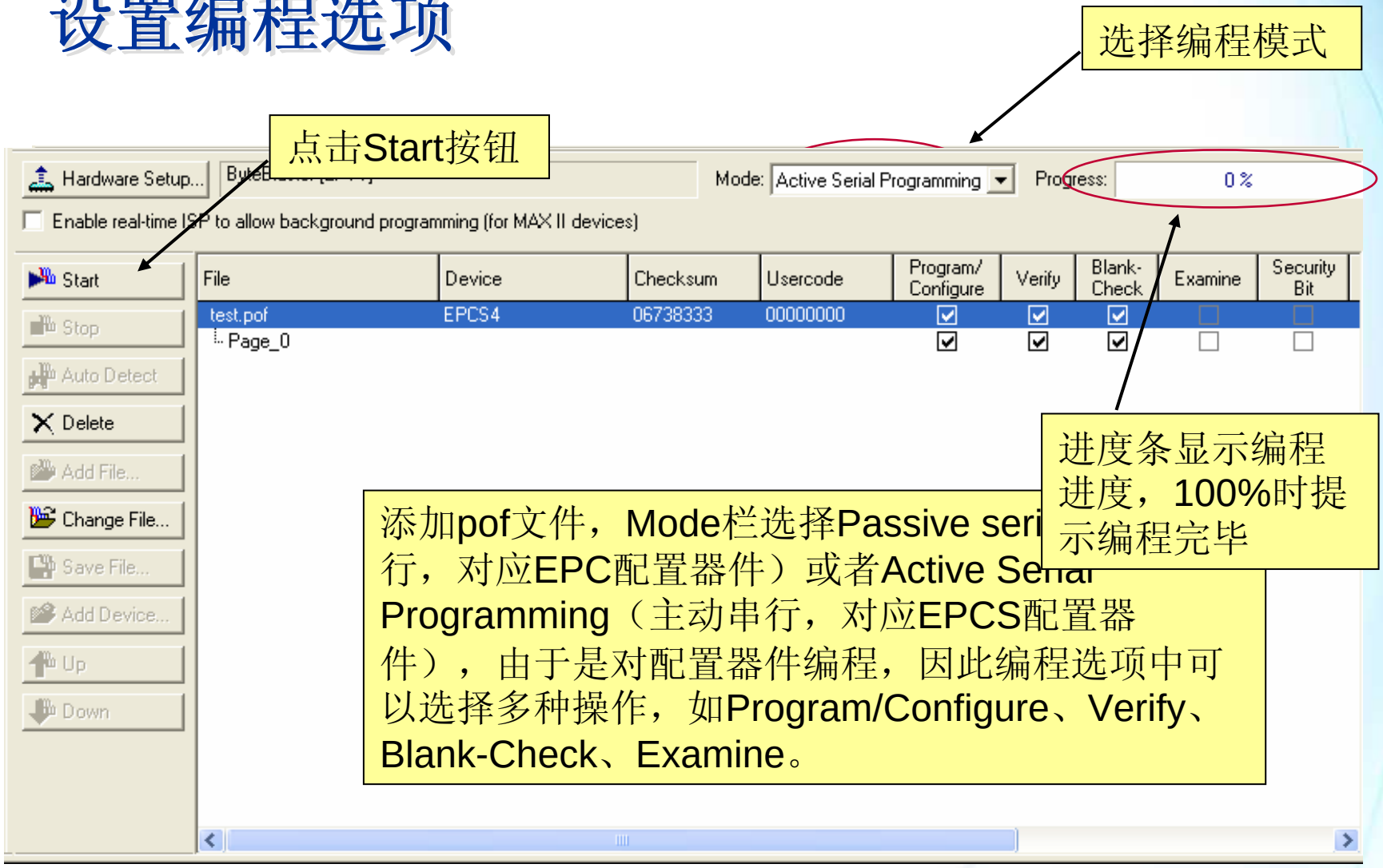
打开编程器窗口

选择Tools-> Programmer或者单击快捷图标，打开编程器窗口



Cytech Technology

设置编程选项

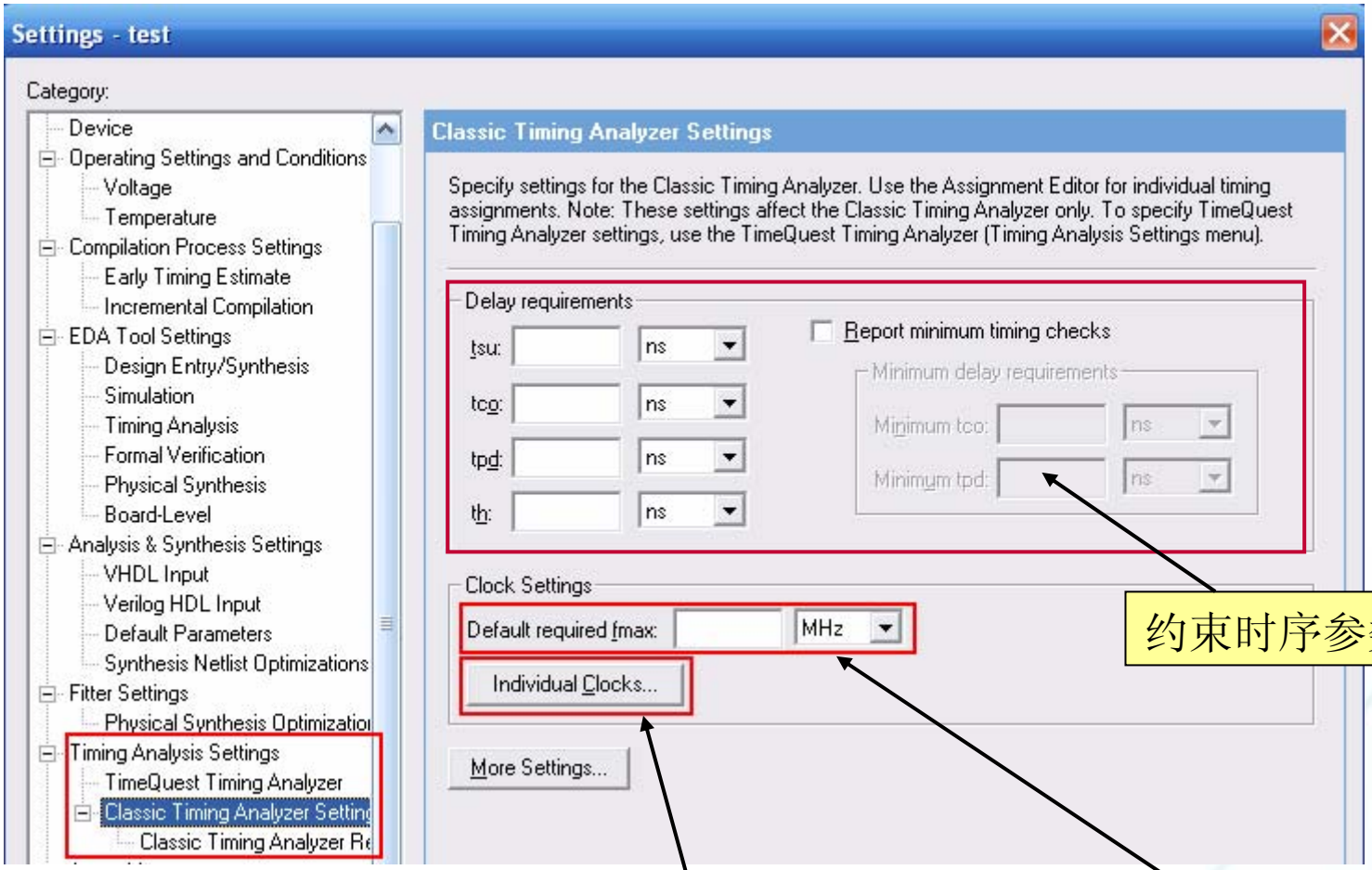


Cytech Technology

Quartus II 软件使用教程

时序约束

时序约束设置—时钟频率设置



-如果设计中的时钟都由PLL产生，QII会自动约束

Cytech Technology

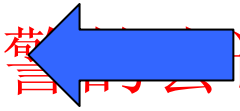
时序分析及优化-最基本的方法

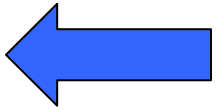
- 时序优化(Fmax优化)最根本、最有效的方式还是对设计代码的优化
- 常用几种速度优化的技巧(Coding Style):
  -增加流水级
  -组合逻辑平衡
  -复制高扇出结点
  -用户状态机设计
  -模块边界输入输出寄存
- QII软件也为工程师提供了很多方便设计优化的选项

Cytech Technology

时序分析及优化-优化首选及保持时间优化

- 通过时序分析报告发现时序存在的问题
  -结合List Path和Locate功能
- 不可以把所有优化选项一起加上，需要对症下药
- 首选第一步是设置综合优化选项，选择优化目标的优先原则
- 对于保持时间告警的尝试解决，设置保持时间的优化选项



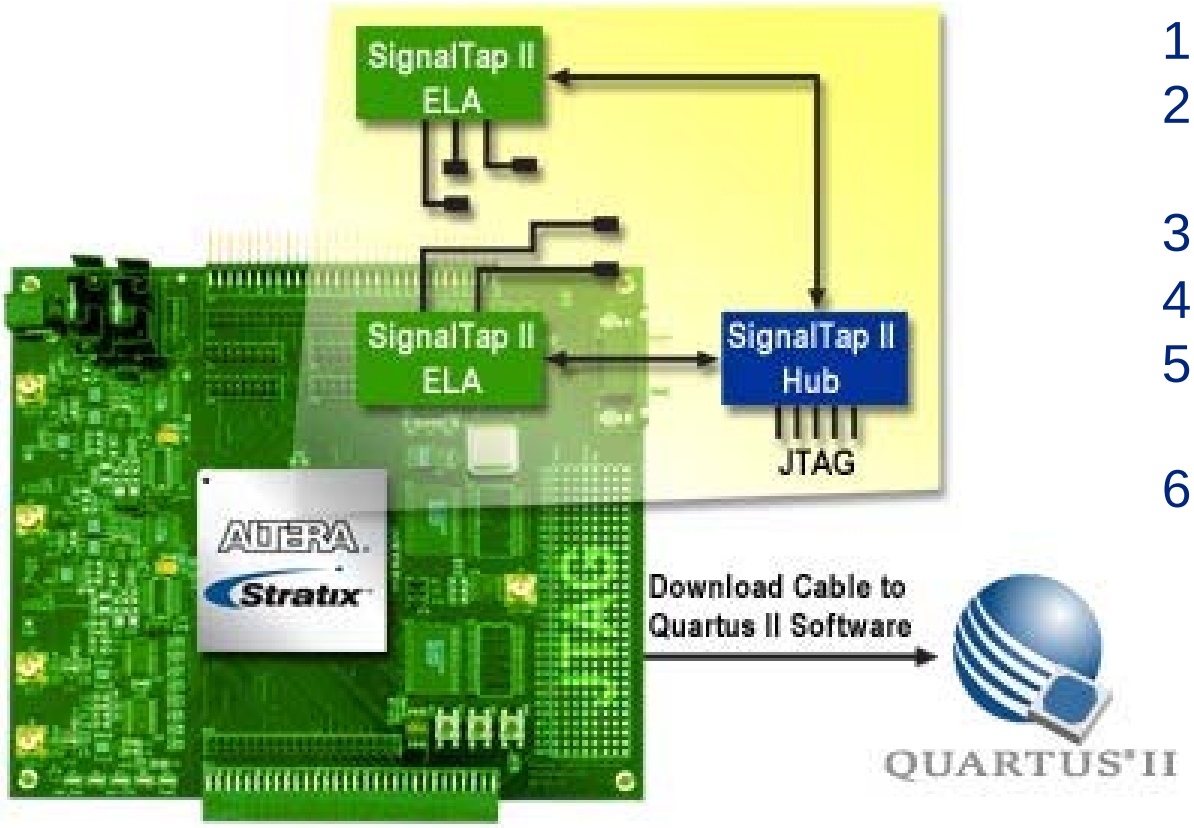


Cytech Technology

Quartus II 软件使用教程

SignalTap II 逻辑分析仪

SignalTap II如何工作?



1. 配置ELA
2. 将ELA和原有设计一起下载到FPGA中
3. 启动 ELA
4. 定义触发条件
5. 采样，并将数据存储到FPGA内部剩余RAM中
6. 通过JTAG口将采样数据传递给Quartus II软件

Cytech Technology

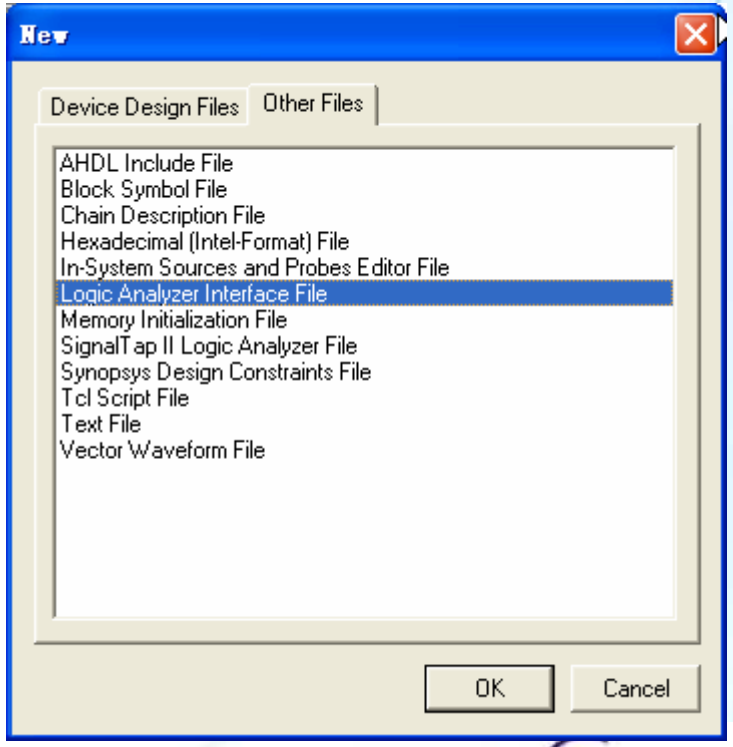
特性

Feature	Benefit
支持多个SignalTap II 核	支持在单个芯片中同时存在多个不同时钟域或者不同功能模块的逻辑分析仪IP函数
支持采样时钟超过200MHz	使用户可以实时观察信号
支持增量编译	允许用户添加/编辑逻辑分析仪的属性，而不影响现有设计的布局布线
最大1024个数据通道/ 每个通道最大128K采样点	允许用户观察大量大量的采样数据
最大10级触发条件	为设置复杂的触发条件提供了足够的灵活性
支持外部触发	允许用户用外部信号触发逻辑分析仪或者输出一个触发信号
支持基本或高级触发功能	支持信号电平、固定值、复杂算数逻辑或者状态机多种触发模式
支持多种文件格式存储数据	使得采样数据可以被第三方验证工具读入、显示和分析

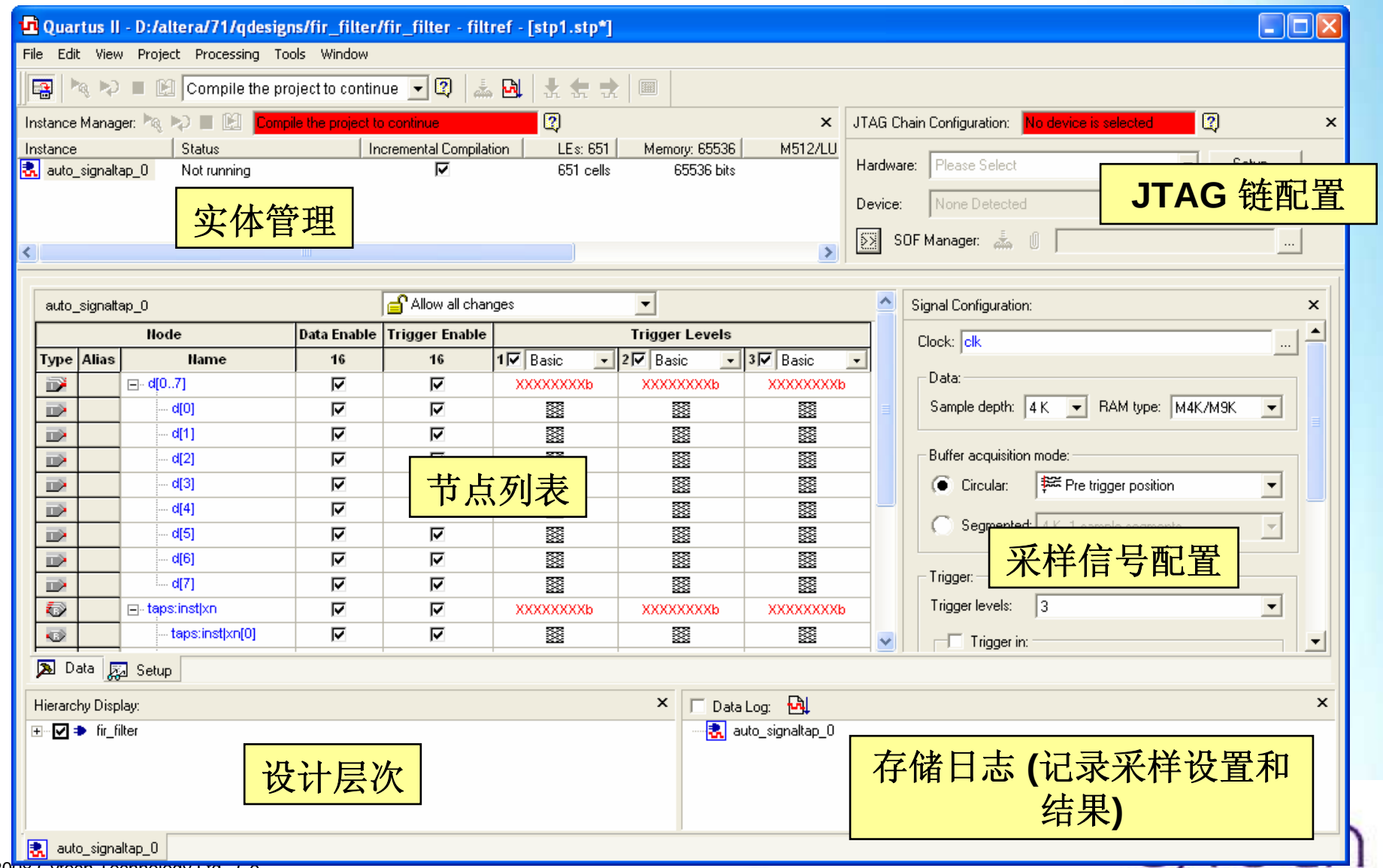
Cytech Technology

1) 创建一个新的 .STP 文件

- 方法1
  - 选择菜单**Tools → SignalTap II Embedded Logic Analyzer**
- 方法2
  - 选择菜单**File → New**
- 默认文件名为 stp1.stp

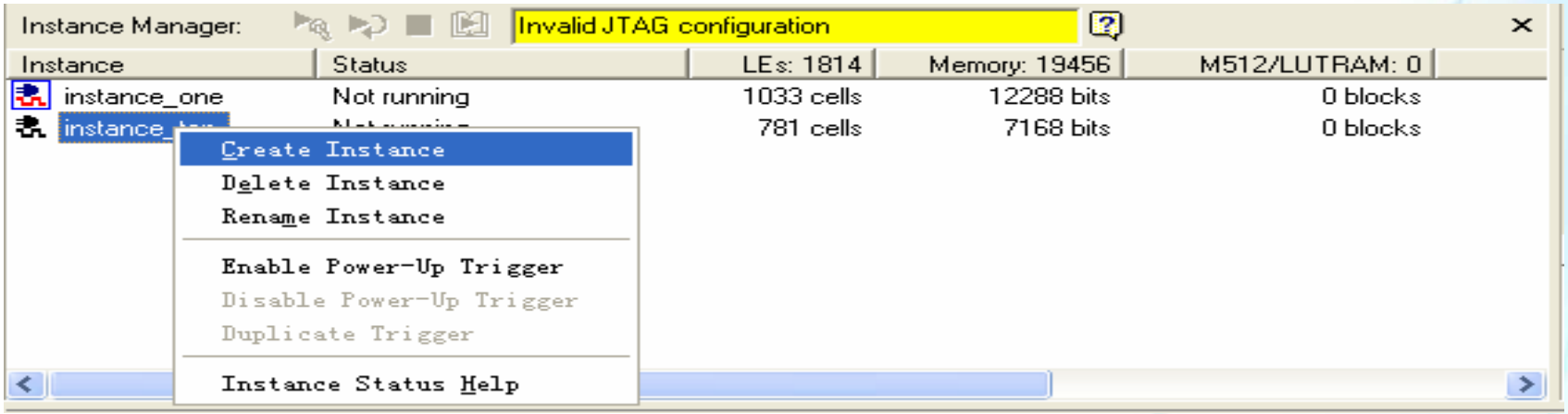


.STP 文件的界面



实体管理

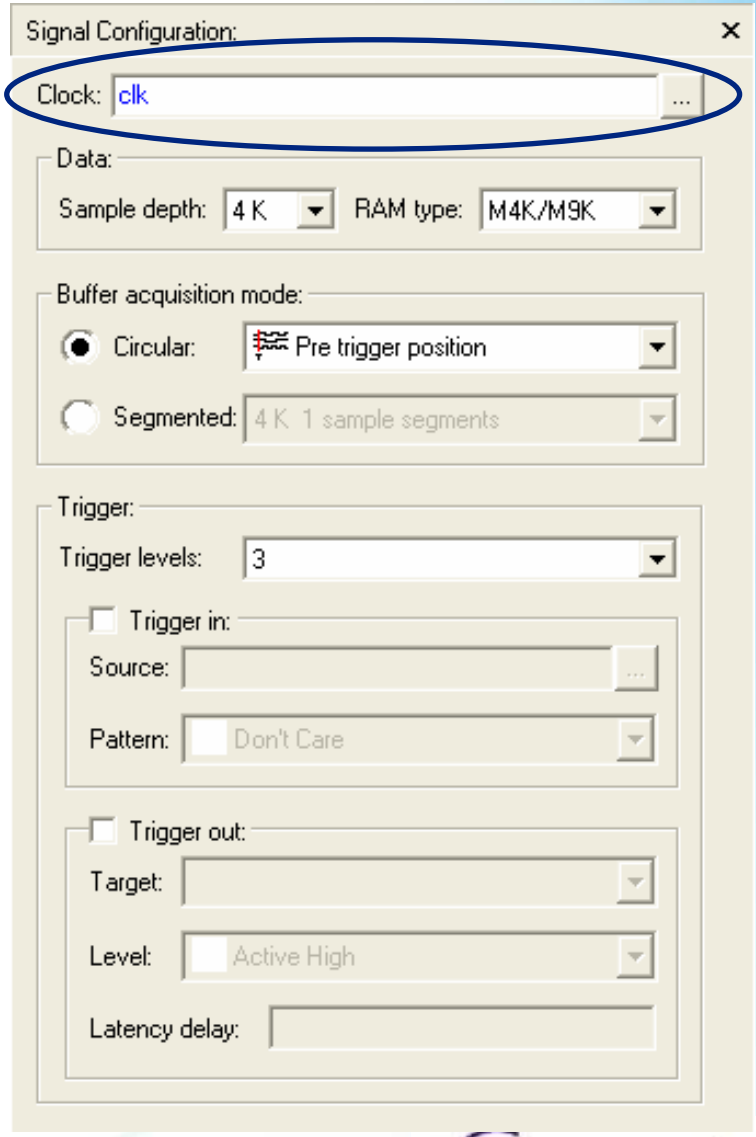
- 增加/删除.STP文件中的实体模块
- 切换对哪个实体进行操作
- 显示ELA占用的资源
- 运行和控制实体



Cytech Technology

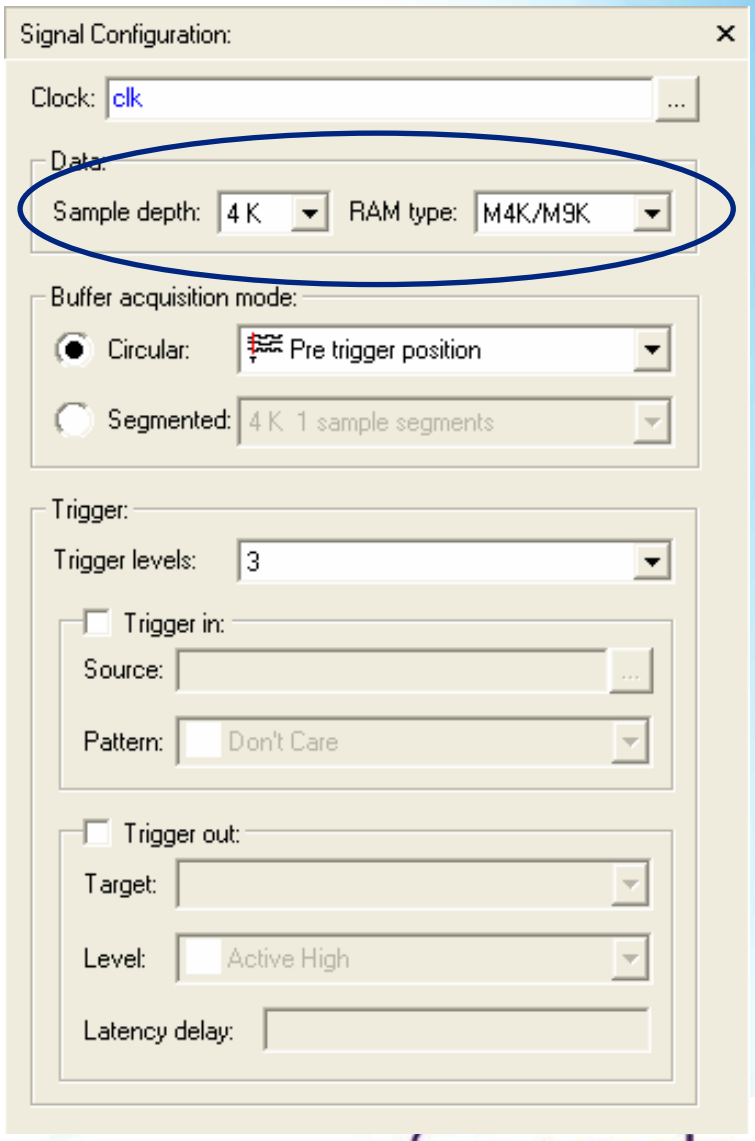
设置采样时钟

- Altera推荐使用全局时钟，而不要使用门控时钟
- 在每个采样时钟上升沿将被测信号存储到缓存
- 如果没有分配采样时钟，软件自动产生一个外部引脚名
  - auto_stp_external_clock
  - ELA 建议此外部信号连接到专用时钟脚上 (用Pin Planner分配)



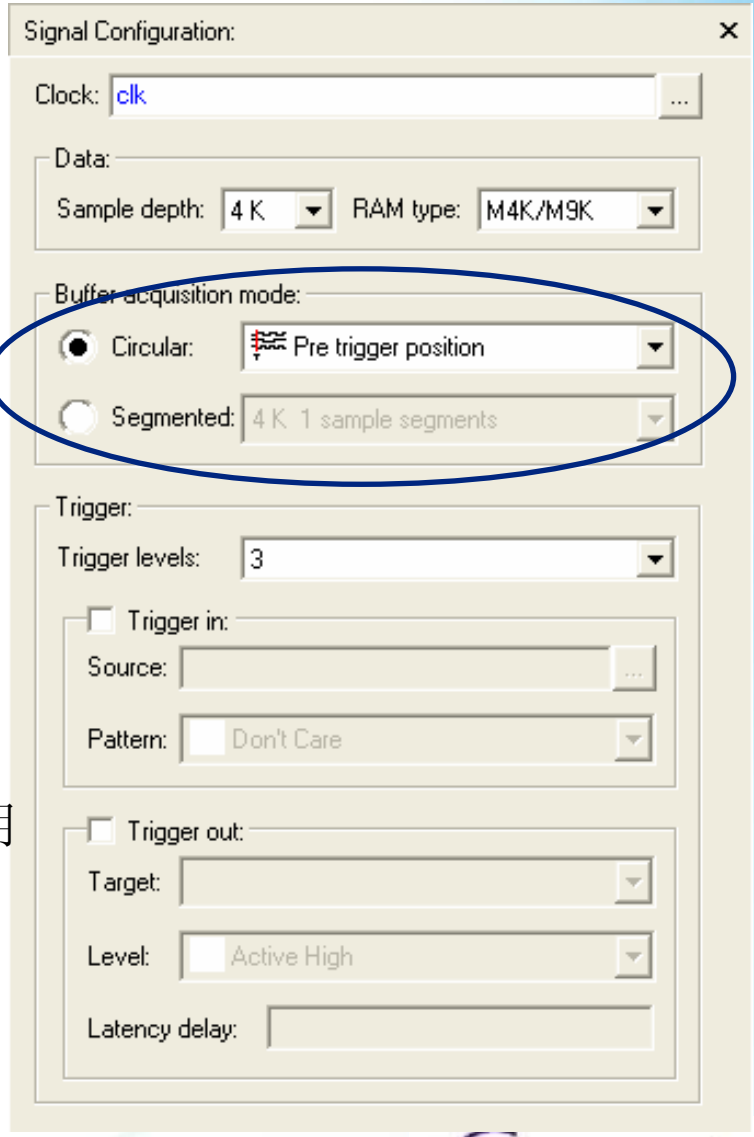
指定采样深度和RAM类型

- 采样深度
  - 设置每个信号的采样点数
  - 0 to 128K 采样深度
- SignalTap II所能显示的被测信号波形的时间长度为Tx，计算公式如下：
  - Tx=N×Ts
  - N为缓存中存储的采样点数，Ts为采样时钟的周期
- 选择RAM类型
  - 选择适当的RAM有利于节省RAM资源

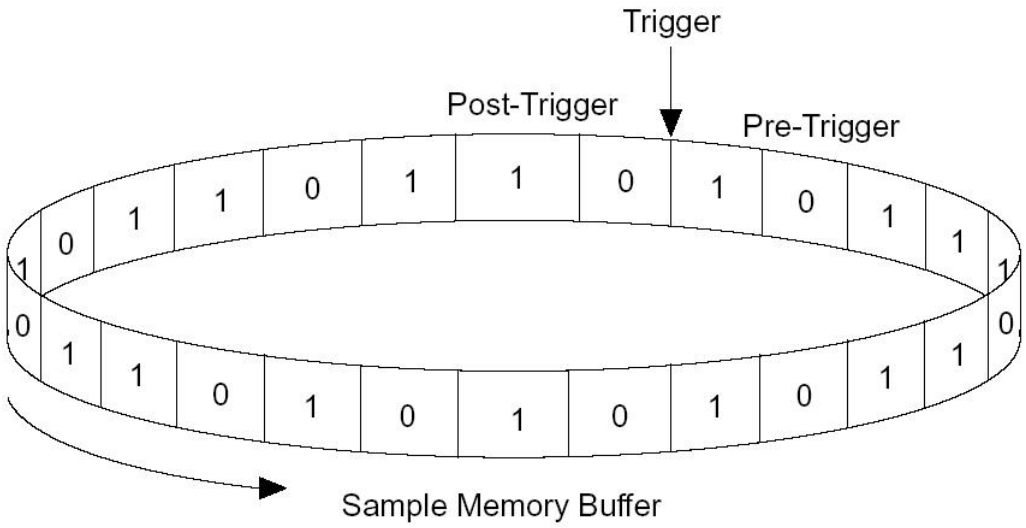


数据获取模式

- 环形存储
  - 指定trigger位置
    - Pre (12% before trigger, 88% after)
    - Center (50% before, 50% after)
    - Post (88% before, 12% after)
    - Continuous
- 分段存储
  - 将整个缓存分成多个片段(segment)，每当触发条件满足时就捕获一段数据。
  - 可以去掉无关的数据，使采样缓存的使用更加灵活

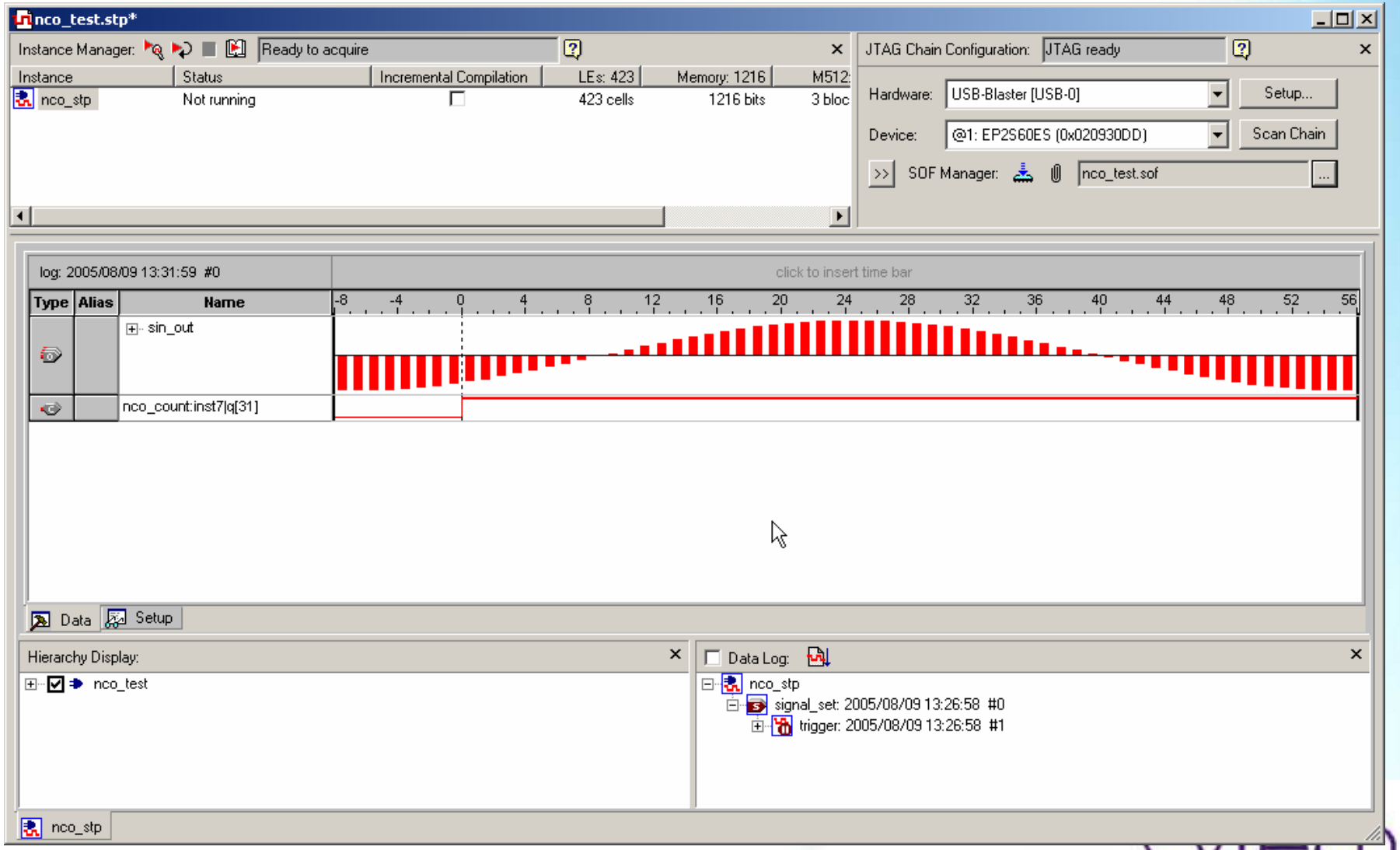


环形Buffer

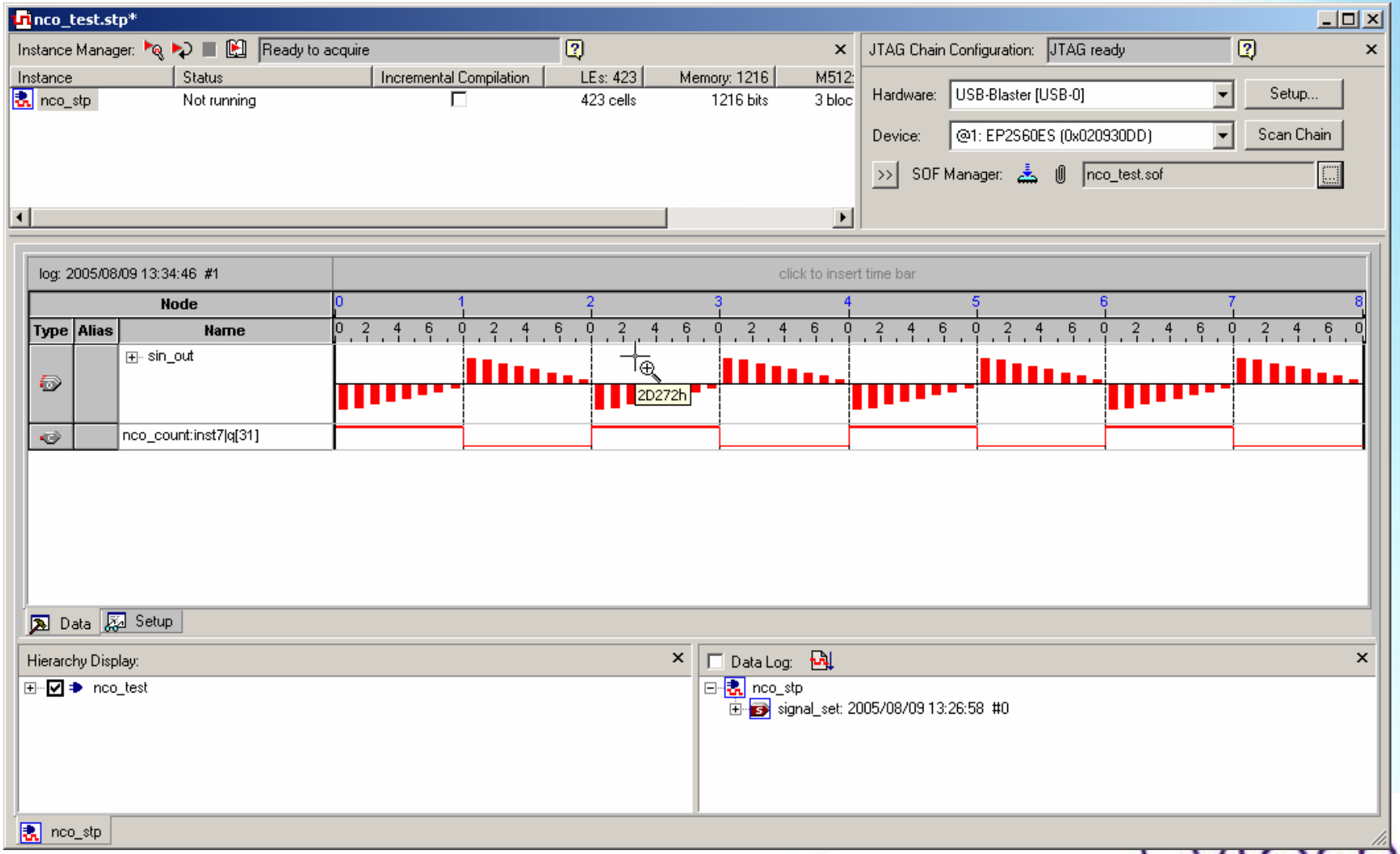


- 触发前，采样数据循环填入一个环形数据缓冲区中
- 触发后，软件采集post-trigger的数据直到填满Buffer

环形Buffer示例

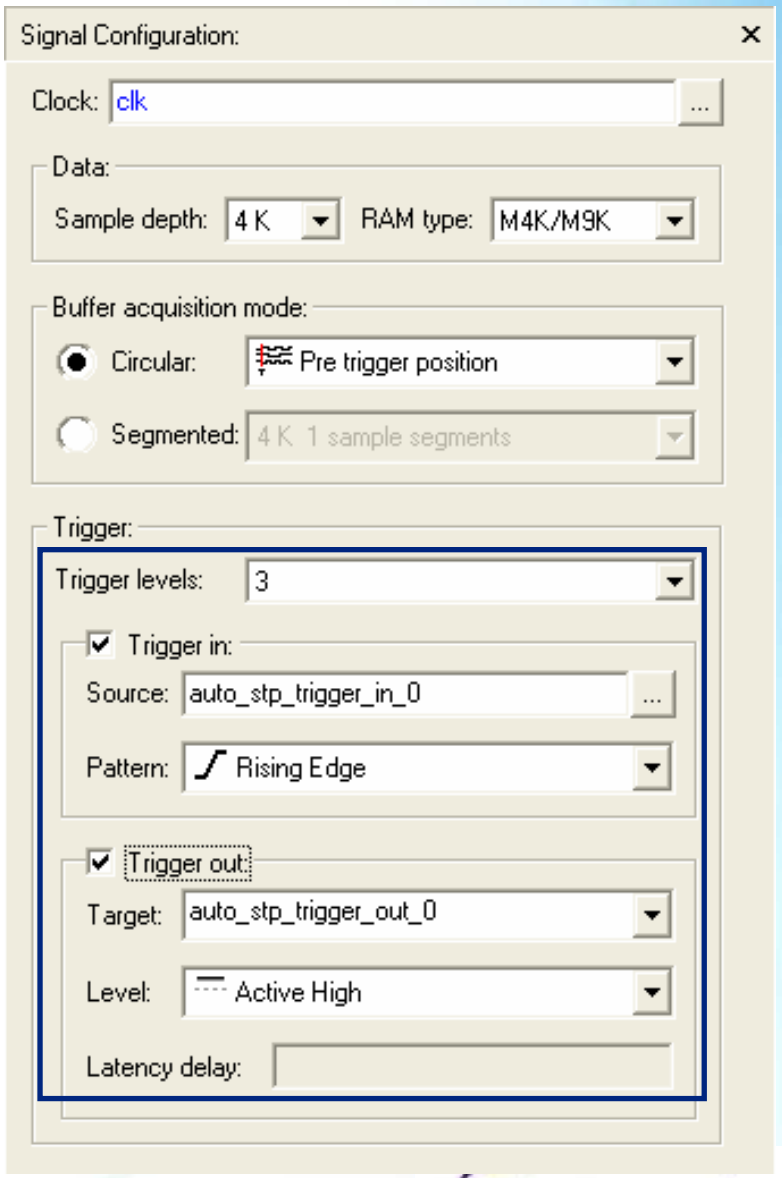


分段Buffer示例

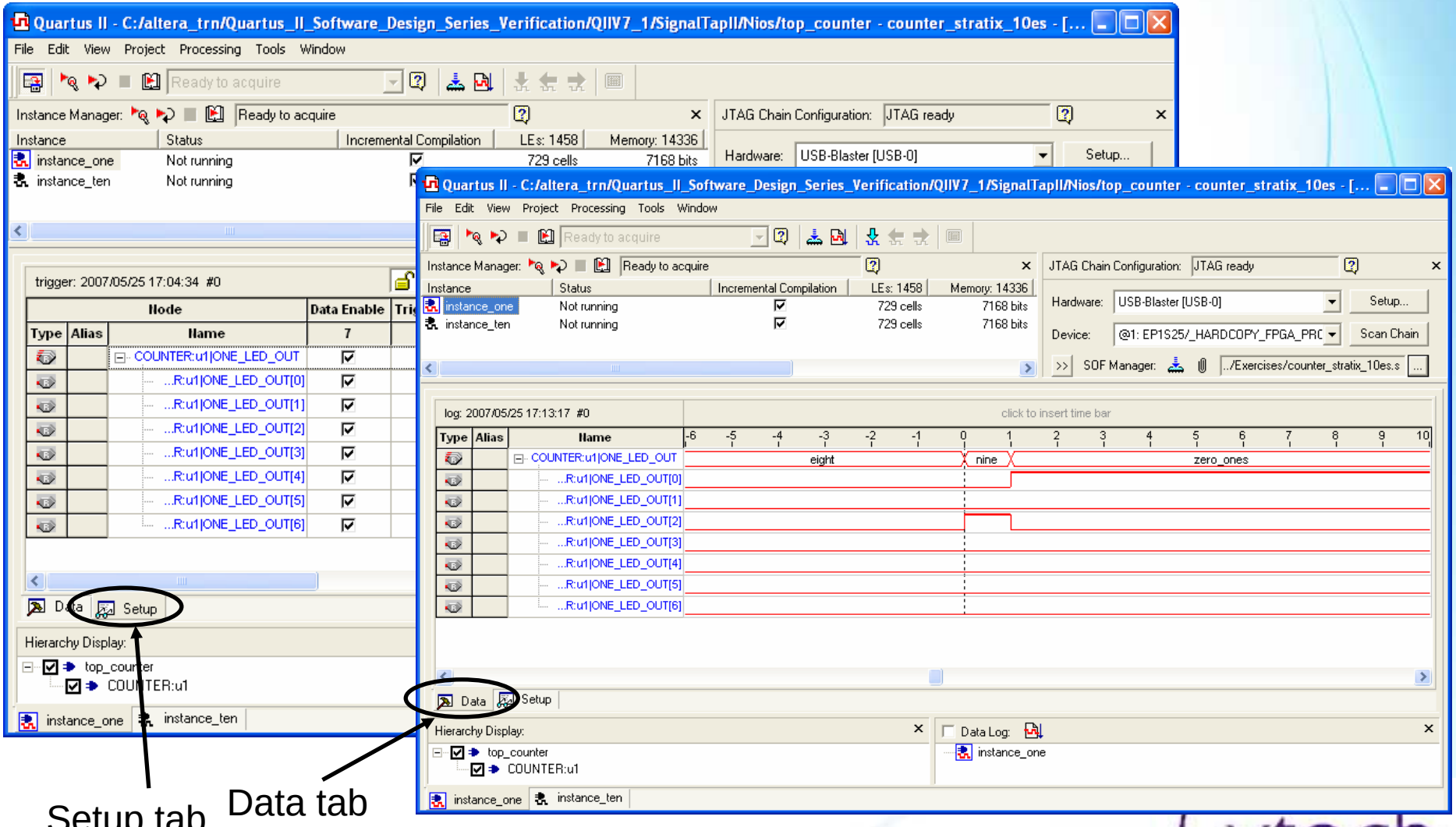


触发

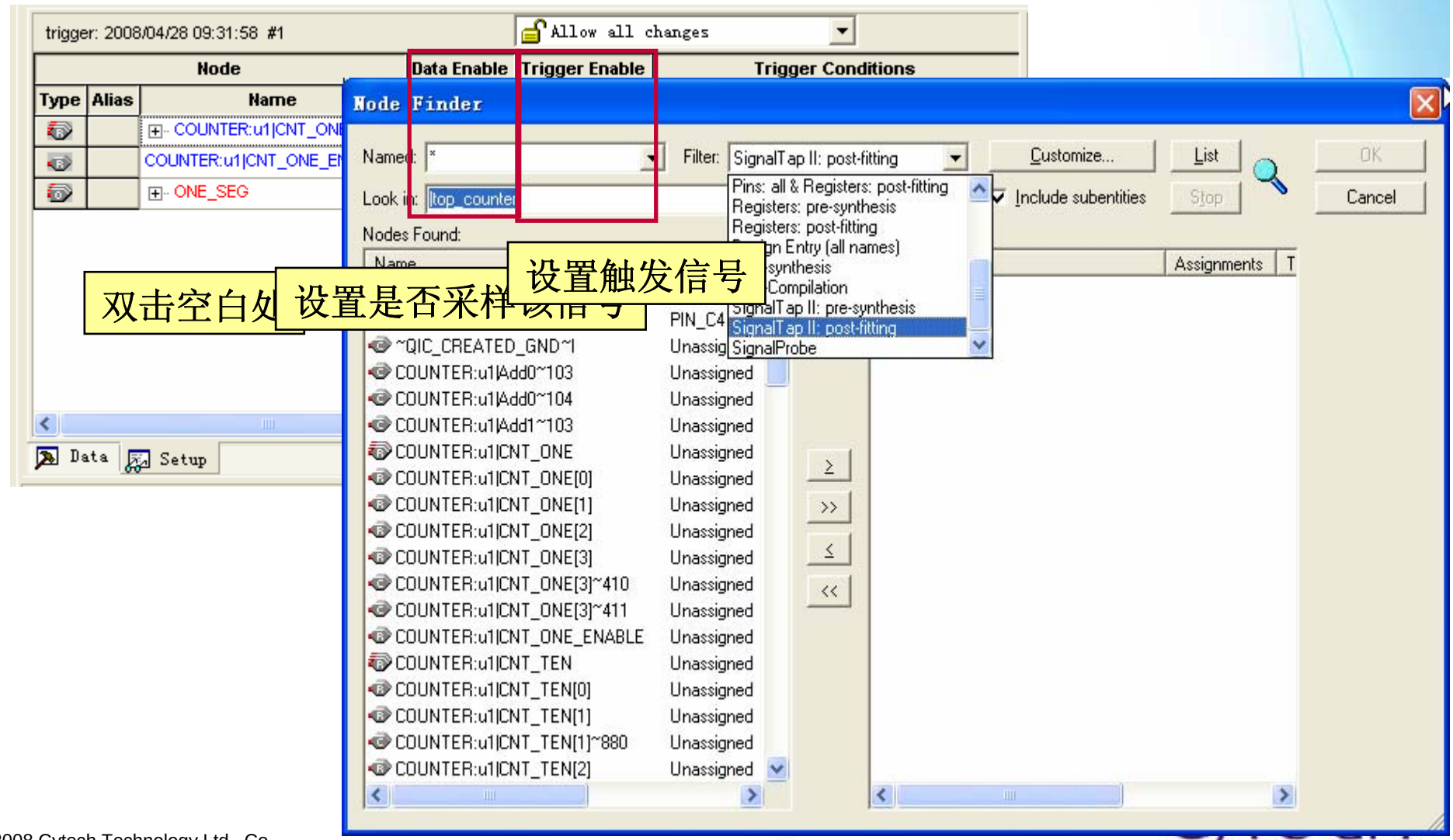
- 触发级别
  - 支持 10 级触发条件
  - 所有事件发生后才开始获取数据
- Trigger-in
  - 任何I/O脚都可以触发分析仪
  - Behaves like trigger level “0”
  - 自动创建auto_stp_trigger_in_n脚，在Pin Planner中进行分配
- Trigger-out
  - 当触发条件满足时，输出一个信号表示触发
  - 自动创建auto_stp_trigger_out_n脚，在Pin Planner中进行分配
    - Latency delay表示从触发到信号输出的延迟时钟数



节点列表 & 波形窗



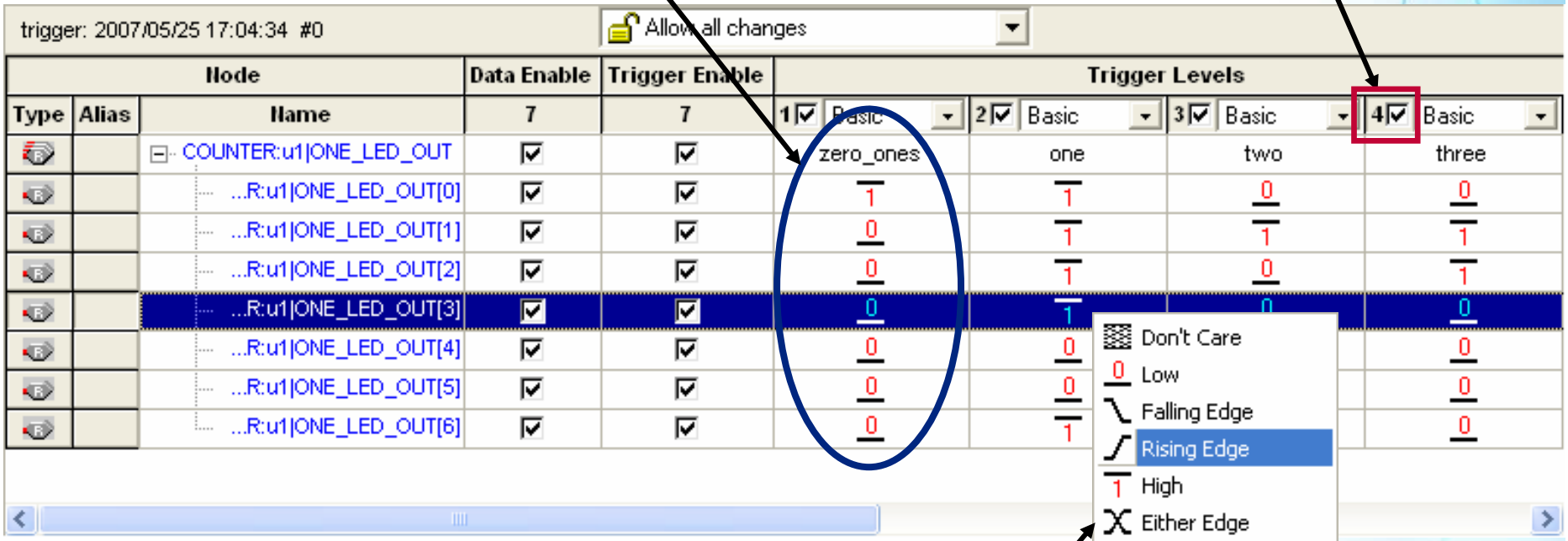
添加节点



基本触发

所有信号都满足时开始获取数据

选择是否使能该级触发条件

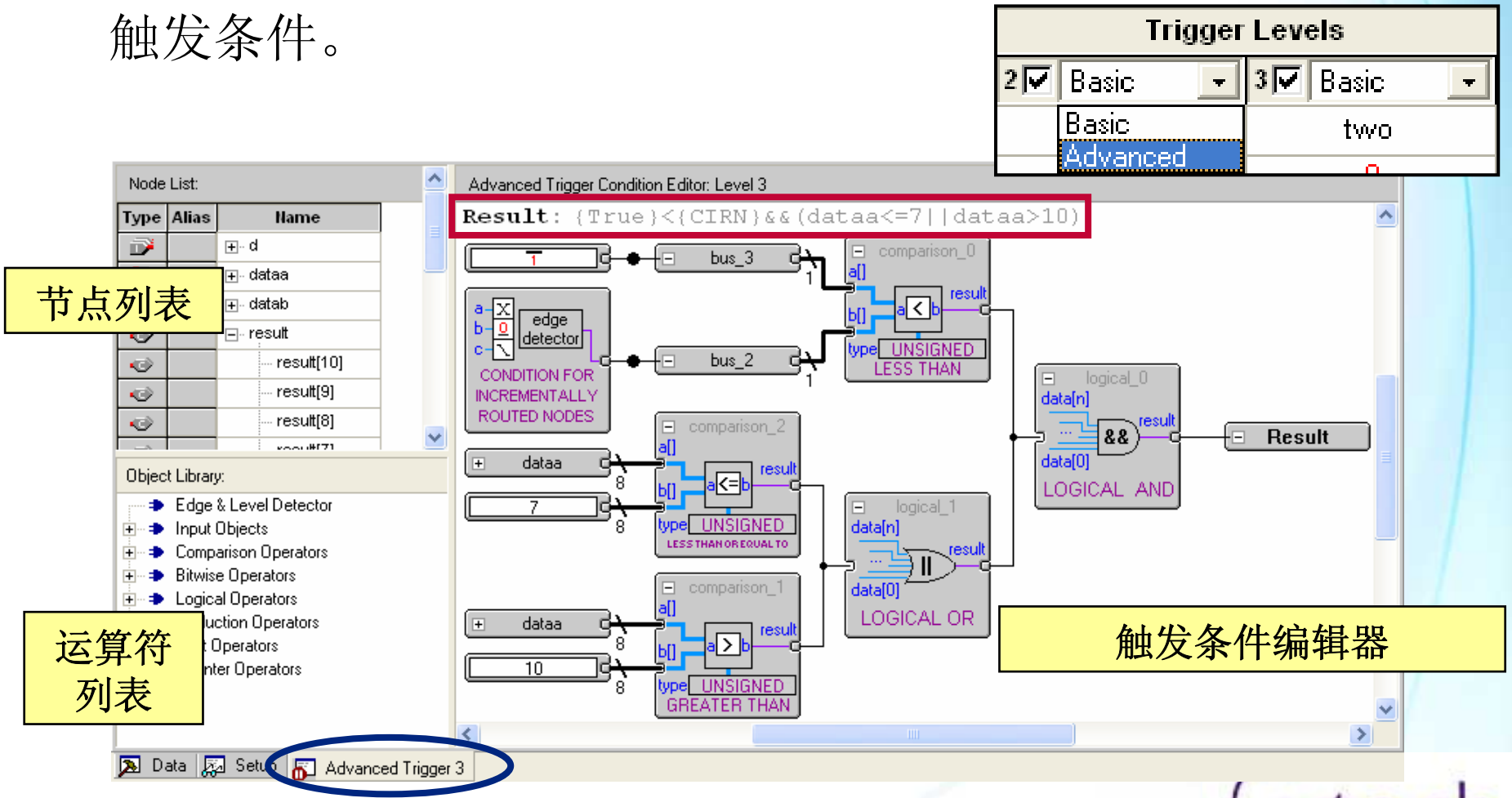


右键设置触发值

Cytech Technology

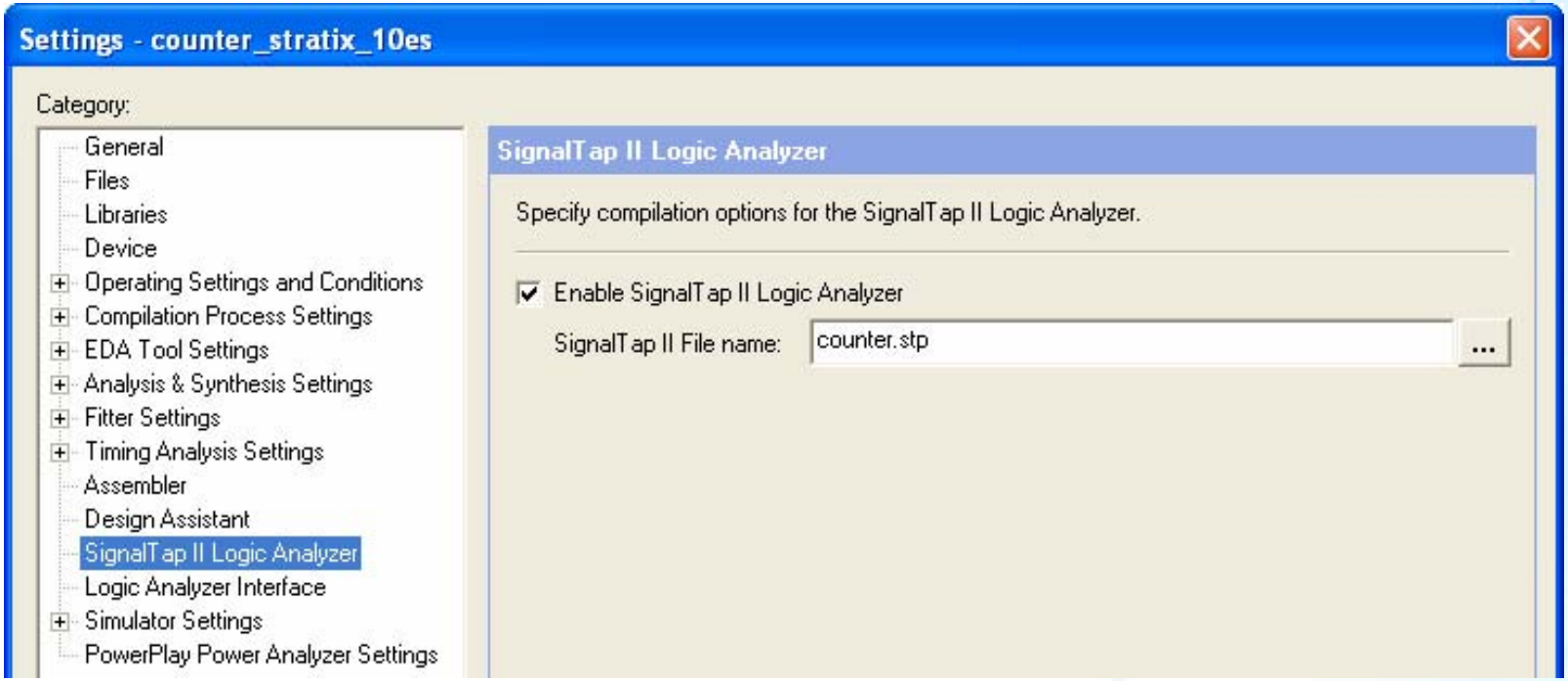
高级触发

- 触发类型选择Advanced时，可以在图形界面中创建复杂的触发条件。



2) 保存.STP 文件 & 编译

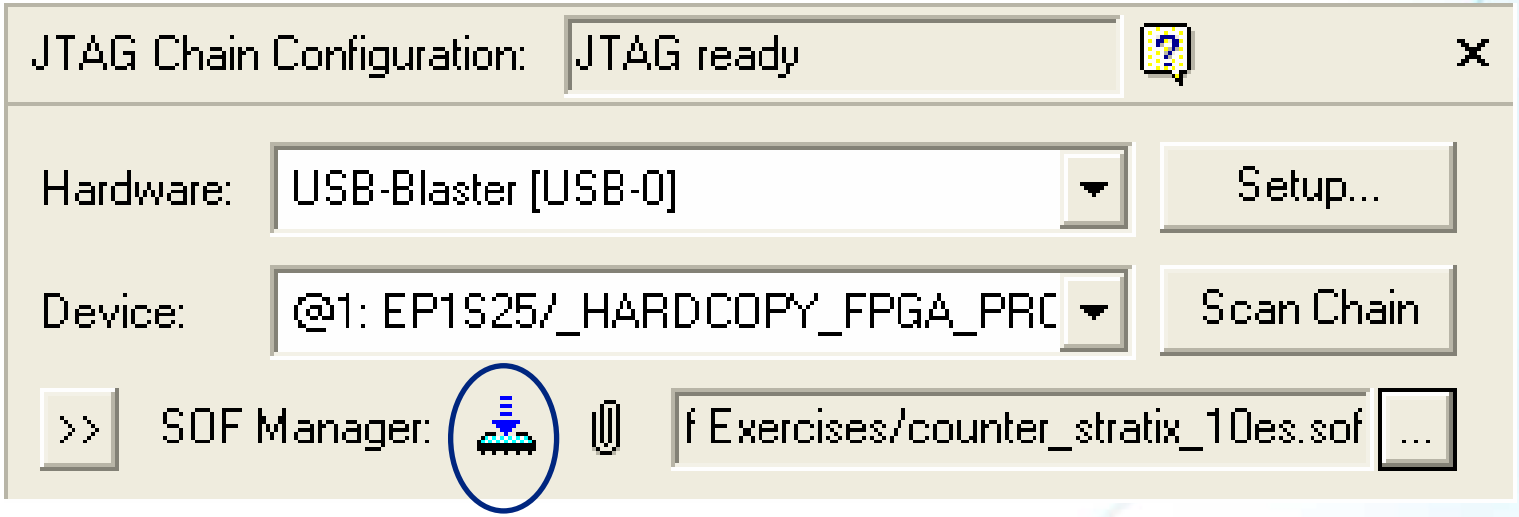
- 选择菜单Assignments → Settings
  - 指定当前使用的SignalTap II文件



Cytech Technology

3) 器件编程

- 使用Quartus II 编程器或者 SignalTap II 分析仪中的编程界面
  - SignalTap II界面中的编程按钮只能对JTAG链中的当前器件进行编程
  - 使用Quartus II 编程器对多个FPGA进行编程
    - 可以为JTAG链上的每个FPGA单独创建一个 SignalTap II 文件

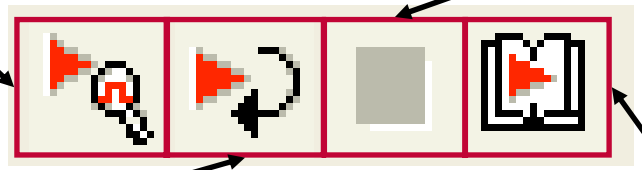


Cytech Technology

4) 获取数据

Run Analysis
单步执行。逻辑分析仪等待触发事件，事件发生时开始采集数据，然后停止

Stop Analysis
停止。如果触发事件还没有发生，则没有接收数据显示



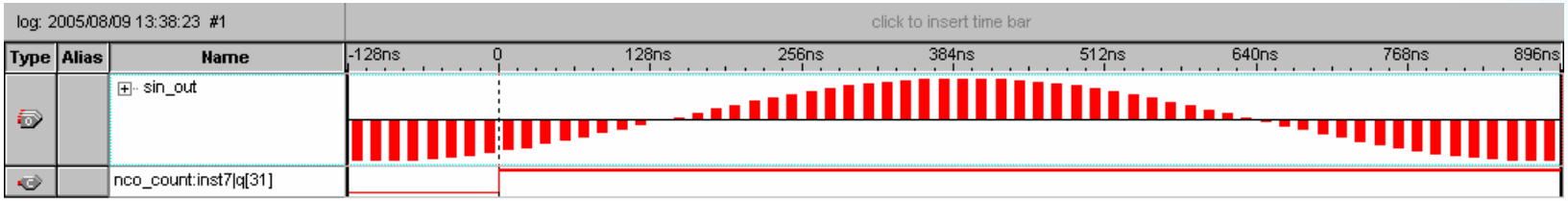
Autorun Analysis
自动运行。逻辑分析仪连续捕获数据，直到用户按下Stop Analysis为止

Read Data
显示捕获的数据。如果触发事件还没有发生，可以点击该按钮察看当前捕获的数据

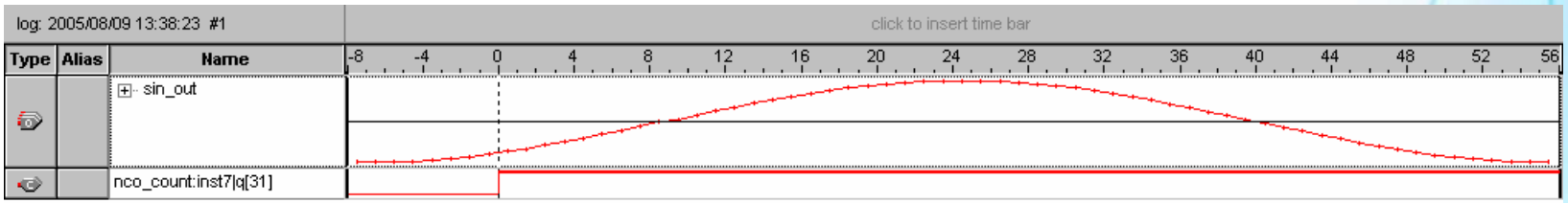
Cytech Technology

显示获取的数据

Time Formatted Bar Graph



Sample Number Formatted Line Graph



- 将
- 创建信号节点的列表文件(菜单File → Create/Update)
- 输出结果供其它验证工具显示和分析 (菜单File → Export)
  - 创建 .VWF, .TBL, .CSV, .VCD, .JPG or .BMP 文件

Cytech Technology

谢 谢！

THANK YOU

Cytech Technology