Yosys Manual

Clifford Wolf

Abstract

Most of today's digital design is done in HDL code (mostly Verilog or VHDL) and with the help of HDL synthesis tools.

In special cases such as synthesis for coarse-grain cell libraries or when testing new synthesis algorithms it might be necessary to write a custom HDL synthesis tool or add new features to an existing one. It this cases the availability of a Free and Open Source (FOSS) synthesis tool that can be used as basis for custom tools would be helpful.

In the absence of such a tool, the Yosys Open SYnthesis Suite (Yosys) was developed. This document covers the design and implementation of this tool. At the moment the main focus of Yosys lies on the high-level aspects of digital synthesis. The pre-existing FOSS logic-synthesis tool ABC is used by Yosys to perform advanced gate-level optimizations.

An evaluation of Yosys based on real-world designs is included. It is shown that Yosys can be used as-is to synthesize such designs. The results produced by Yosys in this tests where successfully verified using formal verification and are comparable in quality to the results produced by a commercial synthesis tool.

This document was originally published as bachelor thesis at the Vienna University of Technology [Wol13].

Abbreviations

AIG	And-Inverter-Graph
ASIC	Application-Specific Integrated Circuit
AST	Abstract Syntax Tree
BDD	Binary Decision Diagram
BLIF	Berkeley Logic Interchange Format
EDA	Electronic Design Automation
EDIF	Electronic Design Interchange Format
ER Diagram	Entity-Relationship Diagram
FOSS	Free and Open-Source Software
FPGA	Field-Programmable Gate Array
FSM	Finite-state machine
HDL	Hardware Description Language
LPM	Library of Parameterized Modules
RTLIL	RTL Intermediate Language
RTL	Register Transfer Level
SAT	Satisfiability Problem
VHDL	VHSIC Hardware Description Language
VHSIC	Very-High-Speed Integrated Circuit
YOSYS	Yosys Open SYnthesis Suite

Contents

CONTENTS

CONTENTS

CONTENTS

CONTENTS

CONTENTS

Chapter 1

Introduction

This document presents the Free and Open Source (FOSS) Verilog HDL synthesis tool "Yosys". Its design and implementation as well as its performance on real-world designs is discussed in this document.

1.1   History of Yosys

A Hardware Description Language (HDL) is a computer language used to describe circuits. A HDL synthesis tool is a computer program that takes a formal description of a circuit written in an HDL as input and generates a netlist that implements the given circuit as output.

Currently the most widely used and supported HDLs for digital circuits are Verilog [Ver06][Ver02] and VHDL[1] [VHD09][VHD04]. Both HDLs are used for test and verification purposes as well as logic synthesis, resulting in a set of synthesizable and a set of non-synthesizable language features. In this document we only look at the synthesizable subset of the language features.

In recent work on heterogeneous coarse-grain reconfigurable logic [WGS$^+$12] the need for a custom application-specific HDL synthesis tool emerged. It was soon realised that a synthesis tool that understood Verilog or VHDL would be preferred over a synthesis tool for a custom HDL. Given an existing Verilog or VHDL front end, the work for writing the necessary additional features and integrating them in an existing tool can be estimated to be about the same as writing a new tool with support for a minimalistic custom HDL.

The proposed custom HDL synthesis tool should be licensed under a Free and Open Source Software (FOSS) licence. So an existing FOSS Verilog or VHDL synthesis tool would have been needed as basis to build upon. The main advantages of choosing Verilog or VHDL is the ability to synthesize existing HDL code and to mitigate the requirement for circuit-designers to learn a new language. In order to take full advantage of any existing FOSS Verilog or VHDL tool, such a tool would have to provide a feature-complete implementation of the synthesizable HDL subset.

Basic RTL synthesis is a well understood field [HS96]. Lexing, parsing and processing of computer languages [ASU86] is a thoroughly researched field. All the information required to write such tools has been openly available for a long time, and it is therefore likely that a FOSS HDL synthesis tool with a feature-complete Verilog or VHDL front end must exist which can be used as a basis for a custom RTL synthesis tool.

Due to the author's preference for Verilog over VHDL it was decided early on to go for Verilog instead of VHDL[2]. So the existing FOSS Verilog synthesis tools were evaluated (see App. **??**). The results of this evaluation are utterly devastating. Therefore a completely new Verilog synthesis tool was implemented and is recommended as basis for custom synthesis tools. This is the tool that is discussed in this document.

---

[1]VHDL is an acronym for "VHSIC hardware description language" and VHSIC is an acronym for "Very-High-Speed Integrated Circuits".
[2]A quick investigation into FOSS VHDL tools yielded similar grim results for FOSS VHDL synthesis tools.

1.2   Structure of this Document

The structure of this document is as follows:

Chapter 1 is this introduction.

Chapter 2 covers a short introduction to the world of HDL synthesis. Basic principles and the terminology are outlined in this chapter.

Chapter 3 gives the quickest possible outline to how the problem of implementing a HDL synthesis tool is approached in the case of Yosys.

Chapter 4 contains a more detailed overview of the implementation of Yosys. This chapter covers the data structures used in Yosys to represent a design in detail and is therefore recommended reading for everyone who is interested in understanding the Yosys internals.

Chapter 5 covers the internal cell library used by Yosys. This is especially important knowledge for anyone who wants to understand the intermediate netlists used internally by Yosys.

Chapter 6 gives a tour to the internal APIs of Yosys. This is recommended reading for everyone who actually wants to read or write Yosys source code. The chapter concludes with an example loadable module for Yosys.

Chapters 7, 8, and 9 cover three important pieces of the synthesis pipeline: The Verilog frontend, the optimization passes and the technology mapping to the target architecture, respectively.

Chapter ?? covers the evaluation of the performance (correctness and quality) of Yosys on real-world input data. The chapter concludes the main part of this document with conclusions and outlook to future work.

Various appendices, including a command reference manual (App. C) and an evaluation of pre-existing FOSS Verilog synthesis tools (App. ??) complete this document.

Chapter 2

Basic Principles

This chapter contains a short introduction to the basic principles of digital circuit synthesis.

2.1 Levels of Abstraction

Digital circuits can be represented at different levels of abstraction. During the design process a circuit is usually first specified using a higher level abstraction. Implementation can then be understood as finding a functionally equivalent representation at a lower abstraction level. When this is done automatically using software, the term *synthesis* is used.

So synthesis is the automatic conversion of a high-level representation of a circuit to a functionally equivalent low-level representation of a circuit. Figure 2.1 lists the different levels of abstraction and how they relate to different kinds of synthesis.

Regardless of the way a lower level representation of a circuit is obtained (synthesis or manual design), the lower level representation is usually verified by comparing simulation results of the lower level and the higher level representation [1]. Therefore even if no synthesis is used, there must still be a simulatable representation of the circuit in all levels to allow for verification of the design.

---

[1] In recent years formal equivalence checking also became an important verification method for validating RTL and lower abstraction representation of the design.
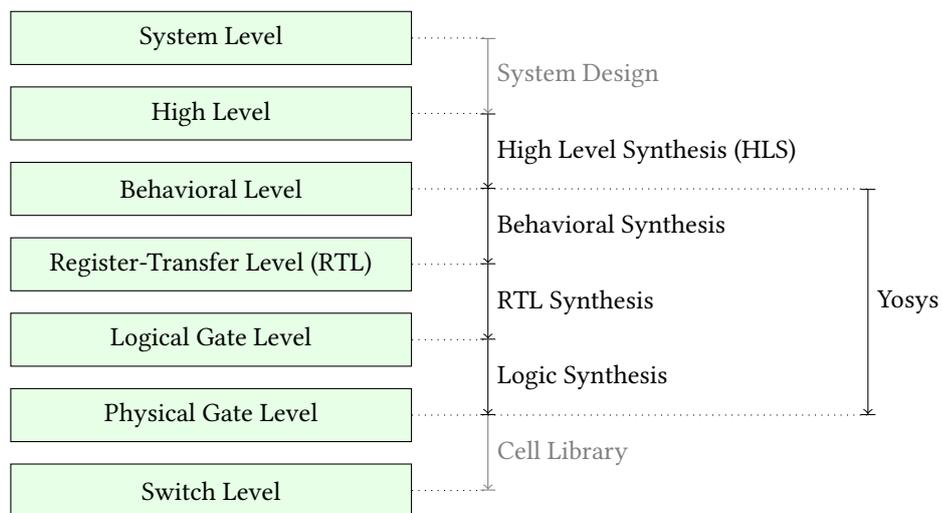
Figure 2.1: Different levels of abstraction and synthesis.

Note: The exact meaning of terminology such as "High-Level" is of course not fixed over time. For example the HDL "ABEL" was first introduced in 1985 as "A High-Level Design Language for Programmable Logic Devices" [LHBB85], but would not be considered a "High-Level Language" today.

2.1.1 System Level

The System Level abstraction of a system only looks at its biggest building blocks like CPUs and computing cores. At this level the circuit is usually described using traditional programming languages like C/C++ or Matlab. Sometimes special software libraries are used that are aimed at simulation circuits on the system level, such as SystemC.

Usually no synthesis tools are used to automatically transform a system level representation of a circuit to a lower-level representation. But system level design tools exist that can be used to connect system level building blocks.

The IEEE 1685-2009 standard defines the IP-XACT file format that can be used to represent designs on the system level and building blocks that can be used in such system level designs. [IP-10]

2.1.2 High Level

The high-level abstraction of a system (sometimes referred to as *algorithmic* level) is also often represented using traditional programming languages, but with a reduced feature set. For example when representing a design at the high level abstraction in C, pointers can only be used to mimic concepts that can be found in hardware, such as memory interfaces. Full featured dynamic memory management is not allowed as it has no corresponding concept in digital circuits.

Tools exist to synthesize high level code (usually in the form of C/C++/SystemC code with additional metadata) to behavioural HDL code (usually in the form of Verilog or VHDL code). Aside from the many commercial tools for high level synthesis there are also a number of FOSS tools for high level synthesis [16] [19].

2.1.3 Behavioural Level

At the behavioural abstraction level a language aimed at hardware description such as Verilog or VHDL is used to describe the circuit, but so-called *behavioural modelling* is used in at least part of the circuit description. In behavioural modelling there must be a language feature that allows for imperative programming to be used to describe data paths and registers. This is the `always`-block in Verilog and the `process`-block in VHDL.

In behavioural modelling, code fragments are provided together with a *sensitivity list*; a list of signals and conditions. In simulation, the code fragment is executed whenever a signal in the sensitivity list changes its value or a condition in the sensitivity list is triggered. A synthesis tool must be able to transfer this representation into an appropriate datapath followed by the appropriate types of register.

For example consider the following Verilog code fragment:

```verilog
1  always @(posedge clk)
2        y <= a + b;
```

In simulation the statement `y <= a + b` is executed whenever a positive edge on the signal `clk` is detected. The synthesis result however will contain an adder that calculates the sum `a + b` all the time, followed by a d-type flip-flop with the adder output on its D-input and the signal `y` on its Q-output.

Usually the imperative code fragments used in behavioural modelling can contain statements for conditional execution (`if`- and `case`-statements in Verilog) as well as loops, as long as those loops can be completely unrolled.

Interestingly there seems to be no other FOSS Tool that is capable of performing Verilog or VHDL behavioural syntheses besides Yosys (see App. ??).

2.1.4 Register-Transfer Level (RTL)

On the Register-Transfer Level the design is represented by combinatorial data paths and registers (usually d-type flip flops). The following Verilog code fragment is equivalent to the previous Verilog example, but is in RTL representation:

```
1  assign tmp = a + b;        // combinatorial data path
2
3  always @(posedge clk)      // register
4          y <= tmp;
```

A design in RTL representation is usually stored using HDLs like Verilog and VHDL. But only a very limited subset of features is used, namely minimalistic `always`-blocks (Verilog) or `process`-blocks (VHDL) that model the register type used and unconditional assignments for the datapath logic. The use of HDLs on this level simplifies simulation as no additional tools are required to simulate a design in RTL representation.

Many optimizations and analyses can be performed best at the RTL level. Examples include FSM detection and optimization, identification of memories or other larger building blocks and identification of shareable resources.

Note that RTL is the first abstraction level in which the circuit is represented as a graph of circuit elements (registers and combinatorial cells) and signals. Such a graph, when encoded as list of cells and connections, is called a netlist.

RTL synthesis is easy as each circuit node element in the netlist can simply be replaced with an equivalent gate-level circuit. However, usually the term *RTL synthesis* does not only refer to synthesizing an RTL netlist to a gate level netlist but also to performing a number of highly sophisticated optimizations within the RTL representation, such as the examples listed above.

A number of FOSS tools exist that can perform isolated tasks within the domain of RTL synthesis steps. But there seems to be no FOSS tool that covers a wide range of RTL synthesis operations.

2.1.5 Logical Gate Level

At the logical gate level the design is represented by a netlist that uses only cells from a small number of single-bit cells, such as basic logic gates (AND, OR, NOT, XOR, etc.) and registers (usually D-Type Flip-flops).

A number of netlist formats exists that can be used on this level, e.g. the Electronic Design Interchange Format (EDIF), but for ease of simulation often a HDL netlist is used. The latter is a HDL file (Verilog or VHDL) that only uses the most basic language constructs for instantiation and connecting of cells.

There are two challenges in logic synthesis: First finding opportunities for optimizations within the gate level netlist and second the optimal (or at least good) mapping of the logic gate netlist to an equivalent netlist of physically available gate types.

The simplest approach to logic synthesis is *two-level logic synthesis*, where a logic function is converted into a sum-of-products representation, e.g. using a Karnaugh map. This is a simple approach, but has exponential worst-case effort and cannot make efficient use of physical gates other than AND/NAND-, OR/NOR- and NOT-Gates.

Therefore modern logic synthesis tools utilize much more complicated *multi-level logic synthesis* algorithms [BHSV90]. Most of these algorithms convert the logic function to a Binary-Decision-Diagram (BDD) or And-Inverter-Graph (AIG) and work from that representation. The former has the advantage that it has a unique normalized form. The latter has much better worst case performance and is therefore better suited for the synthesis of large logic functions.

Good FOSS tools exists for multi-level logic synthesis [27] [26] [28].

Yosys contains basic logic synthesis functionality but can also use ABC [27] for the logic synthesis step. Using ABC is recommended.

2.1.6 Physical Gate Level

On the physical gate level only gates are used that are physically available on the target architecture. In some cases this may only be NAND, NOR and NOT gates as well as D-Type registers. In other cases this might include cells that are more complex than the cells used at the logical gate level (e.g. complete half-adders). In the case of an FPGA-based design the physical gate level representation is a netlist of LUTs with optional output registers, as these are the basic building blocks of FPGA logic cells.

For the synthesis tool chain this abstraction is usually the lowest level. In case of an ASIC-based design the cell library might contain further information on how the physical cells map to individual switches (transistors).

2.1.7 Switch Level

A switch level representation of a circuit is a netlist utilizing single transistors as cells. Switch level modelling is possible in Verilog and VHDL, but is seldom used in modern designs, as in modern digital ASIC or FPGA flows the physical gates are considered the atomic build blocks of the logic circuit.

2.1.8 Yosys

Yosys is a Verilog HDL synthesis tool. This means that it takes a behavioural design description as input and generates an RTL, logical gate or physical gate level description of the design as output. Yosys' main strengths are behavioural and RTL synthesis. A wide range of commands (synthesis passes) exist within Yosys that can be used to perform a wide range of synthesis tasks within the domain of behavioural, rtl and logic synthesis. Yosys is designed to be extensible and therefore is a good basis for implementing custom synthesis tools for specialised tasks.

2.2 Features of Synthesizable Verilog

The subset of Verilog [Ver06] that is synthesizable is specified in a separate IEEE standards document, the IEEE standard 1364.1-2002 [Ver02]. This standard also describes how certain language constructs are to be interpreted in the scope of synthesis.

This section provides a quick overview of the most important features of synthesizable Verilog, structured in order of increasing complexity.

2.2.1 Structural Verilog

Structural Verilog (also known as *Verilog Netlists*) is a Netlist in Verilog syntax. Only the following language constructs are used in this case:

- Constant values

- Wire and port declarations

- Static assignments of signals to other signals

- Cell instantiations

Many tools (especially at the back end of the synthesis chain) only support structural Verilog as input. ABC is an example of such a tool. Unfortunately there is no standard specifying what *Structural Verilog* actually is, leading to some confusion about what syntax constructs are supported in structural Verilog when it comes to features such as attributes or multi-bit signals.

2.2.2 Expressions in Verilog

In all situations where Verilog accepts a constant value or signal name, expressions using arithmetic operations such as +, - and *, boolean operations such as & (AND), | (OR) and ^ (XOR) and many others (comparison operations, unary operator, etc.) can also be used.

During synthesis these operators are replaced by cells that implement the respective function.

Many FOSS tools that claim to be able to process Verilog in fact only support basic structural Verilog and simple expressions. Yosys can be used to convert full featured synthesizable Verilog to this simpler subset, thus enabling such applications to be used with a richer set of Verilog features.

2.2.3 Behavioural Modelling

Code that utilizes the Verilog `always` statement is using *Behavioural Modelling*. In behavioural modelling, a circuit is described by means of imperative program code that is executed on certain events, namely any change, a rising edge, or a falling edge of a signal. This is a very flexible construct during simulation but is only synthesizable when one of the following is modelled:

- **Asynchronous or latched logic**
  In this case the sensitivity list must contain all expressions that are used within the `always` block. The syntax `@*` can be used for these cases. Examples of this kind include:

```verilog
1  // asynchronous
2  always @* begin
3          if (add_mode)
4                  y <= a + b;
5          else
6                  y <= a - b;
7  end
8
9  // latched
10 always @* begin
11         if (!hold)
12                 y <= a + b;
13 end
```

  Note that latched logic is often considered bad style and in many cases just the result of sloppy HDL design. Therefore many synthesis tools generate warnings whenever latched logic is generated.

- **Synchronous logic (with optional synchronous reset)**
  This is logic with d-type flip-flops on the output. In this case the sensitivity list must only contain the respective clock edge. Example:

```verilog
1  // counter with synchronous reset
2  always @(posedge clk) begin
3          if (reset)
4                  y <= 0;
5          else
6                  y <= y + 1;
7  end
```

- **Synchronous logic with asynchronous reset**
  This is logic with d-type flip-flops with asynchronous resets on the output. In this case the sensitivity list must only contain the respective clock and reset edges. The values assigned in the reset branch must be constant. Example:

```
1  // counter with asynchronous reset
2  always @(posedge clk, posedge reset) begin
3          if (reset)
4                  y <= 0;
5          else
6                  y <= y + 1;
7  end
```

Many synthesis tools support a wider subset of flip-flops that can be modelled using `always`-statements (including Yosys). But only the ones listed above are covered by the Verilog synthesis standard and when writing new designs one should limit herself or himself to these cases.

In behavioural modelling, blocking assignments (=) and non-blocking assignments (<=) can be used. The concept of blocking vs. non-blocking assignment is one of the most misunderstood constructs in Verilog [CI00].

The blocking assignment behaves exactly like an assignment in any imperative programming language, while with the non-blocking assignment the right hand side of the assignment is evaluated immediately but the actual update of the left hand side register is delayed until the end of the time-step. For example the Verilog code `a <= b; b <= a;` exchanges the values of the two registers. See Sec. **??** for a more detailed description of this behaviour.

2.2.4 Functions and Tasks

Verilog supports *Functions* and *Tasks* to bundle statements that are used in multiple places (similar to *Procedures* in imperative programming). Both constructs can be implemented easily by substituting the function/task-call with the body of the function or task.

2.2.5 Conditionals, Loops and Generate-Statements

Verilog supports `if-else`-statements and `for`-loops inside `always`-statements.

It also supports both features in `generate`-statements on the module level. This can be used to selectively enable or disable parts of the module based on the module parameters (`if-else`) or to generate a set of similar subcircuits (`for`).

While the `if-else`-statement inside an always-block is part of behavioural modelling, the three other cases are (at least for a synthesis tool) part of a built-in macro processor. Therefore it must be possible for the synthesis tool to completely unroll all loops and evaluate the condition in all `if-else`-statement in `generate`-statements using const-folding.

Examples for this can be found in Fig. **??** and Fig. **??** in App. **??**.

2.2.6 Arrays and Memories

Verilog supports arrays. This is in general a synthesizable language feature. In most cases arrays can be synthesized by generating addressable memories. However, when complex or asynchronous access patterns are used, it is not possible to model an array as memory. In these cases the array must be modelled using individual signals for each word and all accesses to the array must be implemented using large multiplexers.

In some cases it would be possible to model an array using memories, but it is not desired. Consider the following delay circuit:

```verilog
1  module (clk, in_data, out_data);
2
3  parameter BITS = 8;
4  parameter STAGES = 4;
5
6  input clk;
7  input [BITS-1:0] in_data;
8  output [BITS-1:0] out_data;
9  reg [BITS-1:0] ffs [STAGES-1:0];
10
11 integer i;
12 always @(posedge clk) begin
13         ffs[0] <= in_data;
14         for (i = 1; i < STAGES; i = i+1)
15                 ffs[i] <= ffs[i-1];
16 end
17
18 assign out_data = ffs[STAGES-1];
19
20 endmodule
```

This could be implemented using an addressable memory with STAGES input and output ports. A better implementation would be to use a simple chain of flip-flops (a so-called shift register). This better implementation can either be obtained by first creating a memory-based implementation and then optimizing it based on the static address signals for all ports or directly identifying such situations in the language front end and converting all memory accesses to direct accesses to the correct signals.

2.3   Challenges in Digital Circuit Synthesis

This section summarizes the most important challenges in digital circuit synthesis. Tools can be characterized by how well they address these topics.

2.3.1   Standards Compliance

The most important challenge is compliance with the HDL standards in question (in case of Verilog the IEEE Standards 1364.1-2002 and 1364-2005). This can be broken down in two items:

- Completeness of implementation of the standard

- Correctness of implementation of the standard

Completeness is mostly important to guarantee compatibility with existing HDL code. Once a design has been verified and tested, HDL designers are very reluctant regarding changes to the design, even if it is only about a few minor changes to work around a missing feature in a new synthesis tool.

Correctness is crucial. In some areas this is obvious (such as correct synthesis of basic behavioural models). But it is also crucial for the areas that concern minor details of the standard, such as the exact rules for handling signed expressions, even when the HDL code does not target different synthesis tools. This is because (unlike software source code that is only processed by compilers), in most design flows HDL code is not only processed by the synthesis tool but also by one or more simulators and sometimes even a formal verification tool. It is key for this verification process that all these tools use the same interpretation for the HDL code.
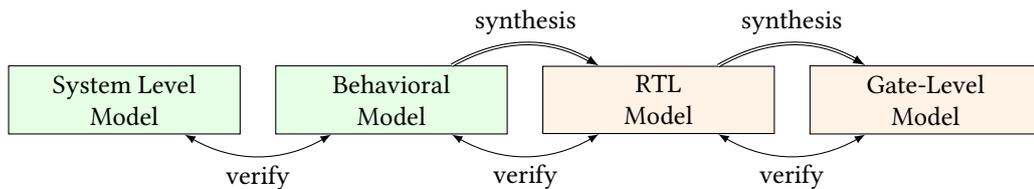
Figure 2.2: Typical design flow. Green boxes represent manually created models. Orange boxes represent models generated by synthesis tools.

2.3.2 Optimizations

Generally it is hard to give a one-dimensional description of how well a synthesis tool optimizes the design. First of all because not all optimizations are applicable to all designs and all synthesis tasks. Some optimizations work (best) on a coarse-grained level (with complex cells such as adders or multipliers) and others work (best) on a fine-grained level (single bit gates). Some optimizations target area and others target speed. Some work well on large designs while others don't scale well and can only be applied to small designs.

A good tool is capable of applying a wide range of optimizations at different levels of abstraction and gives the designer control over which optimizations are performed (or skipped) and what the optimization goals are.

2.3.3 Technology Mapping

Technology mapping is the process of converting the design into a netlist of cells that are available in the target architecture. In an ASIC flow this might be the process-specific cell library provided by the fab. In an FPGA flow this might be LUT cells as well as special function units such as dedicated multipliers. In a coarse-grain flow this might even be more complex special function units.

An open and vendor independent tool is especially of interest if it supports a wide range of different types of target architectures.

2.4 Script-Based Synthesis Flows

A digital design is usually started by implementing a high-level or system-level simulation of the desired function. This description is then manually transformed (or re-implemented) into a synthesizable lower-level description (usually at the behavioural level) and the equivalence of the two representations is verified by simulating both and comparing the simulation results.

Then the synthesizable description is transformed to lower-level representations using a series of tools and the results are again verified using simulation. This process is illustrated in Fig. 2.2.

In this example the System Level Model and the Behavioural Model are both manually written design files. After the equivalence of system level model and behavioural model has been verified, the lower level representations of the design can be generated using synthesis tools. Finally the RTL Model and the Gate-Level Model are verified and the design process is finished.

However, in any real-world design effort there will be multiple iterations for this design process. The reason for this can be the late change of a design requirement or the fact that the analysis of a low-abstraction model (e.g. gate-level timing analysis) revealed that a design change is required in order to meet the design requirements (e.g. maximum possible clock speed).

Whenever the behavioural model or the system level model is changed their equivalence must be re-verified by re-running the simulations and comparing the results. Whenever the behavioural model is changed the synthesis must be re-run and the synthesis results must be re-verified.

Token-Type	Token-Value
`TOK_ASSIGN`	-
`TOK_IDENTIFIER`	"`foo`"
`TOK_EQ`	-
`TOK_IDENTIFIER`	"`bar`"
`TOK_PLUS`	-
`TOK_NUMBER`	42
`TOK_SEMICOLON`	-

Table 2.1: Exemplary token list for the statement "**`assign`** `foo = bar + 42;`".

In order to guarantee reproducibility it is important to be able to re-run all automatic steps in a design project with a fixed set of settings easily. Because of this, usually all programs used in a synthesis flow can be controlled using scripts. This means that all functions are available via text commands. When such a tool provides a GUI, this is complementary to, and not instead of, a command line interface.

Usually a synthesis flow in an UNIX/Linux environment would be controlled by a shell script that calls all required tools (synthesis and simulation/verification in this example) in the correct order. Each of these tools would be called with a script file containing commands for the respective tool. All settings required for the tool would be provided by these script files so that no manual interaction would be necessary. These script files are considered design sources and should be kept under version control just like the source code of the system level and the behavioural model.

2.5   Methods from Compiler Design

Some parts of synthesis tools involve problem domains that are traditionally known from compiler design. This section addresses some of these domains.

2.5.1   Lexing and Parsing

The best known concepts from compiler design are probably *lexing* and *parsing*. These are two methods that together can be used to process complex computer languages easily. [ASU86]

A *lexer* consumes single characters from the input and generates a stream of *lexical tokens* that consist of a *type* and a *value*. For example the Verilog input "**`assign`** `foo = bar + 42;`" might be translated by the lexer to the list of lexical tokens given in Tab. 2.1.

The lexer is usually generated by a lexer generator (e.g. `flex` [17]) from a description file that is using regular expressions to specify the text pattern that should match the individual tokens.

The lexer is also responsible for skipping ignored characters (such as whitespace outside string constants and comments in the case of Verilog) and converting the original text snippet to a token value.

Note that individual keywords use different token types (instead of a keyword type with different token values). This is because the parser usually can only use the Token-Type to make a decision on the grammatical role of a token.

The parser then transforms the list of tokens into a parse tree that closely resembles the productions from the computer languages grammar. As the lexer, the parser is also typically generated by a code generator (e.g. `bison` [18]) from a grammar description in Backus-Naur Form (BNF).

Let's consider the following BNF (in Bison syntax):

```
1  assign_stmt: TOK_ASSIGN TOK_IDENTIFIER TOK_EQ expr TOK_SEMICOLON;
2  expr: TOK_IDENTIFIER | TOK_NUMBER | expr TOK_PLUS expr;
```

The parser converts the token list to the parse tree in Fig. 2.3. Note that the parse tree never actually exists as a whole as data structure in memory. Instead the parser calls user-specified code snippets (so-called *reduce-functions*) for all inner nodes of the parse tree in depth-first order.

In some very simple applications (e.g. code generation for stack machines) it is possible to perform the task at hand directly in the reduce functions. But usually the reduce functions are only used to build an in-memory data structure with the relevant information from the parse tree. This data structure is called an *abstract syntax tree* (AST).

The exact format for the abstract syntax tree is application specific (while the format of the parse tree and token list are mostly dictated by the grammar of the language at hand). Figure 2.4 illustrates what an AST for the parse tree in Fig. 2.3 could look like.

Usually the AST is then converted into yet another representation that is more suitable for further processing. In compilers this is often an assembler-like three-address-code intermediate representation. [ASU86]

2.5.2 Multi-Pass Compilation

Complex problems are often best solved when split up into smaller problems. This is certainly true for compilers as well as for synthesis tools. The components responsible for solving the smaller problems can be connected in two different ways: through *Single-Pass Pipelining* and by using *Multiple Passes*.

Traditionally a parser and lexer are connected using the pipelined approach: The lexer provides a function that is called by the parser. This function reads data from the input until a complete lexical token has been read. Then this token is returned to the parser. So the lexer does not first generate a complete list of lexical tokens and then pass it to the parser. Instead they run concurrently and the parser can consume tokens as the lexer produces them.

The single-pass pipelining approach has the advantage of lower memory footprint (at no time must the complete design be kept in memory) but has the disadvantage of tighter coupling between the interacting components.

Therefore single-pass pipelining should only be used when the lower memory footprint is required or the components are also conceptually tightly coupled. The latter certainly is the case for a parser and its lexer. But when data is passed between two conceptually loosely coupled components it is often beneficial to use a multi-pass approach.

In the multi-pass approach the first component processes all the data and the result is stored in a in-memory data structure. Then the second component is called with this data. This reduces complexity, as only one component is running at a time. It also improves flexibility as components can be exchanged easier.
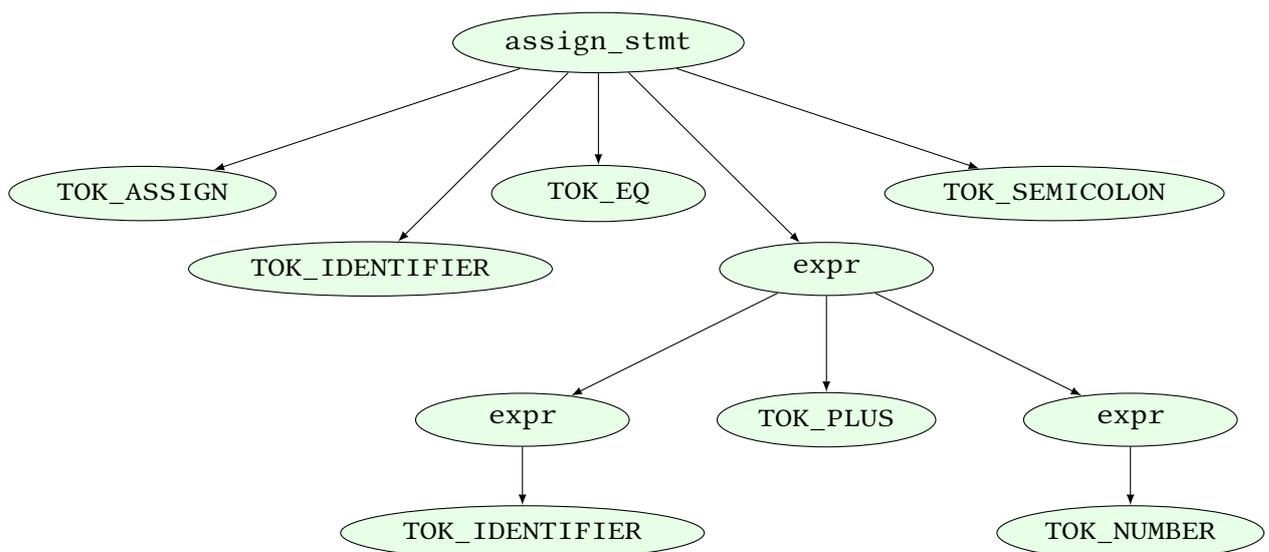
Figure 2.3: Example parse tree for the Verilog expression "**assign** foo = bar + 42;".
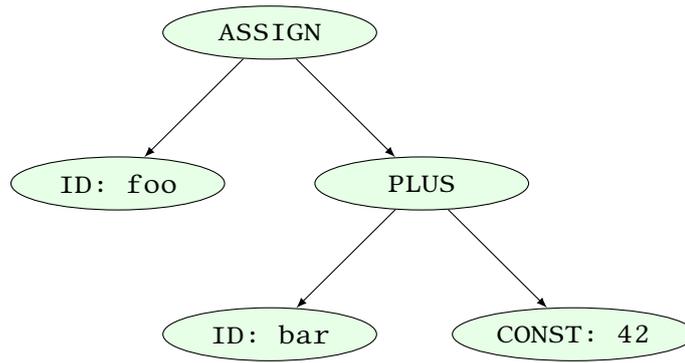
Figure 2.4: Example abstract syntax tree for the Verilog expression "**assign** foo = bar + 42;".

Most modern compilers are multi-pass compilers.

Chapter 3

Approach

Yosys is a tool for synthesising (behavioural) Verilog HDL code to target architecture netlists. Yosys aims at a wide range of application domains and thus must be flexible and easy to adapt to new tasks. This chapter covers the general approach followed in the effort to implement this tool.

3.1 Data- and Control-Flow

The data- and control-flow of a typical synthesis tool is very similar to the data- and control-flow of a typical compiler: different subsystems are called in a predetermined order, each consuming the data generated by the last subsystem and generating the data for the next subsystem (see Fig. 3.1).

The first subsystem to be called is usually called a *frontend*. It does not process the data generated by another subsystem but instead reads the user input—in the case of a HDL synthesis tool, the behavioural HDL code.

The subsystems that consume data from previous subsystems and produce data for the next subsystems (usually in the same or a similar format) are called *passes*.

The last subsystem that is executed transforms the data generated by the last pass into a suitable output format and writes it to a disk file. This subsystem is usually called the *backend*.

In Yosys all frontends, passes and backends are directly available as commands in the synthesis script. Thus the user can easily create a custom synthesis flow just by calling passes in the right order in a synthesis script.
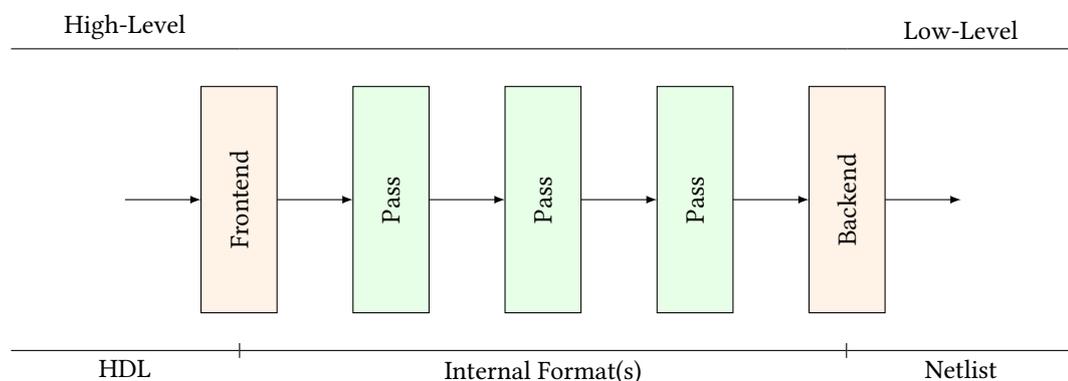
Figure 3.1: General data- and control-flow of a synthesis tool

3.2 Internal Formats in Yosys

Yosys uses two different internal formats. The first is used to store an abstract syntax tree (AST) of a Verilog input file. This format is simply called *AST* and is generated by the Verilog Frontend. This data structure is consumed by a subsystem called *AST Frontend*[1]. This AST Frontend then generates a design in Yosys' main internal format, the Register-Transfer-Level-Intermediate-Language (RTLIL) representation. It does that by first performing a number of simplifications within the AST representation and then generating RTLIL from the simplified AST data structure.

The RTLIL representation is used by all passes as input and outputs. This has the following advantages over using different representational formats between different passes:

- The passes can be rearranged in a different order and passes can be removed or inserted.

- Passes can simply pass-thru the parts of the design they don't change without the need to convert between formats. In fact Yosys passes output the same data structure they received as input and performs all changes in place.

- All passes use the same interface, thus reducing the effort required to understand a pass when reading the Yosys source code, e.g. when adding additional features.

The RTLIL representation is basically a netlist representation with the following additional features:

- An internal cell library with fixed-function cells to represent RTL datapath and register cells as well as logical gate-level cells (single-bit gates and registers).

- Support for multi-bit values that can use individual bits from wires as well as constant bits to represent coarse-grain netlists.

- Support for basic behavioural constructs (if-then-else structures and multi-case switches with a sensitivity list for updating the outputs).

- Support for multi-port memories.

The use of RTLIL also has the disadvantage of having a very powerful format between all passes, even when doing gate-level synthesis where the more advanced features are not needed. In order to reduce complexity for passes that operate on a low-level representation, these passes check the features used in the input RTLIL and fail to run when unsupported high-level constructs are used. In such cases a pass that transforms the higher-level constructs to lower-level constructs must be called from the synthesis script first.

3.3 Typical Use Case

The following example script may be used in a synthesis flow to convert the behavioural Verilog code from the input file `design.v` to a gate-level netlist `synth.v` using the cell library described by the Liberty file [25] `cells.lib`:

```
1  # read input file to internal representation
2  read_verilog design.v
3
4  # convert high-level behavioral parts ("processes") to d-type flip-flops and muxes
5  proc
6
7  # perform some simple optimizations
```

---

[1]In Yosys the term *pass* is only used to refer to commands that operate on the RTLIL data structure.

```
 8 │ opt
 9 │
10 │ # convert high-level memory constructs to d-type flip-flops and multiplexers
11 │ memory
12 │
13 │ # perform some simple optimizations
14 │ opt
15 │
16 │ # convert design to (logical) gate-level netlists
17 │ techmap
18 │
19 │ # perform some simple optimizations
20 │ opt
21 │
22 │ # map internal register types to the ones from the cell library
23 │ dfflibmap -liberty cells.lib
24 │
25 │ # use ABC to map remaining logic to cells from the cell library
26 │ abc -liberty cells.lib
27 │
28 │ # cleanup
29 │ opt
30 │
31 │ # write results to output file
32 │ write_verilog synth.v
```

A detailed description of the commands available in Yosys can be found in App. C.

Chapter 4

Implementation Overview

Yosys is an extensible open source hardware synthesis tool. It is aimed at designers who are looking for an easily accessible, universal, and vendor-independent synthesis tool, as well as scientists who do research in electronic design automation (EDA) and are looking for an open synthesis framework that can be used to test algorithms on complex real-world designs.

Yosys can synthesize a large subset of Verilog 2005 and has been tested with a wide range of real-world designs, including the OpenRISC 1200 CPU [23], the openMSP430 CPU [22], the OpenCores I^{2}C master [20] and the k68 CPU [21].

As of this writing a Yosys VHDL frontend is in development.

Yosys is written in C++ (using some features from the new C++11 standard). This chapter describes some of the fundamental Yosys data structures. For the sake of simplicity the C++ type names used in the Yosys implementation are used in this chapter, even though the chapter only explains the conceptual idea behind it and can be used as reference to implement a similar system in any language.

4.1   Simplified Data Flow

Figure 4.1 shows the simplified data flow within Yosys. Rectangles in the figure represent program modules and ellipses internal data structures that are used to exchange design data between the program modules.

Design data is read in using one of the frontend modules. The high-level HDL frontends for Verilog and VHDL code generate an abstract syntax tree (AST) that is then passed to the AST frontend. Note that both HDL frontends use the same AST representation that is powerful enough to cover the Verilog HDL and VHDL language.

The AST Frontend then compiles the AST to Yosys's main internal data format, the RTL Intermediate Language (RTLIL). A more detailed description of this format is given in the next section.

There is also a text representation of the RTLIL data structure that can be parsed using the ILANG Frontend.

The design data may then be transformed using a series of passes that all operate on the RTLIL representation of the design.

Finally the design in RTLIL representation is converted back to text by one of the backends, namely the Verilog Backend for generating Verilog netlists and the ILANG Backend for writing the RTLIL data in the same format that is understood by the ILANG Frontend.

With the exception of the AST Frontend, which is called by the high-level HDL frontends and can't be called directly by the user, all program modules are called by the user (usually using a synthesis script that contains text commands for Yosys).
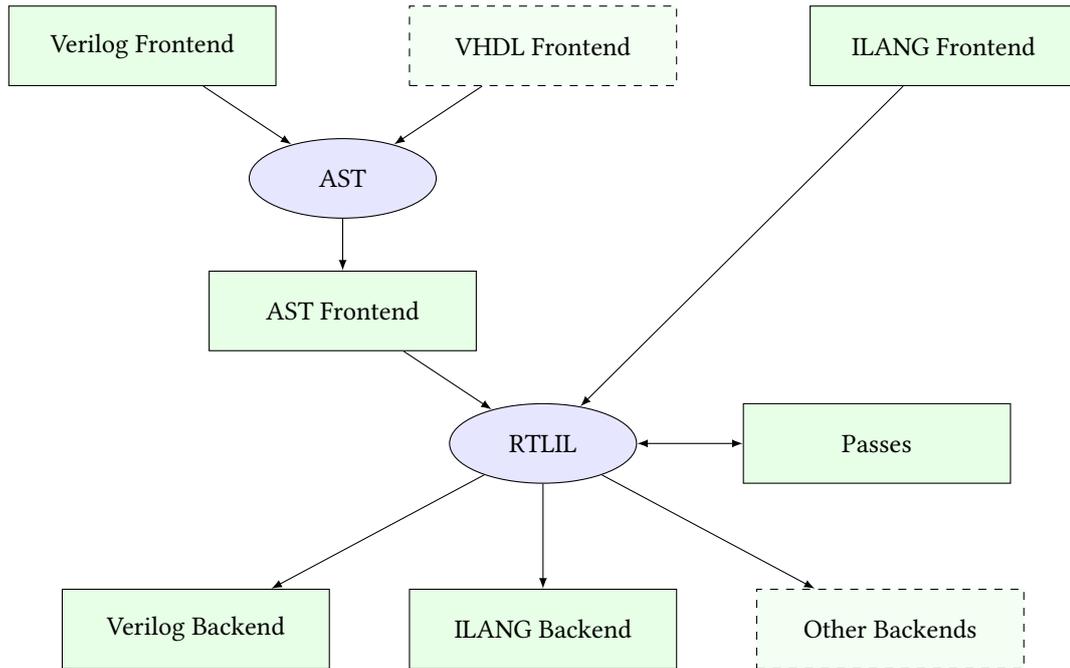
Figure 4.1: Yosys simplified data flow (ellipses: data structures, rectangles: program modules)

By combining passes in different ways and/or adding additional passes to Yosys it is possible to adapt Yosys to a wide range of applications. For this to be possible it is key that (1) all passes operate on the same data structure (RTLIL) and (2) that this data structure is powerful enough to represent the design in different stages of the synthesis.

4.2 The RTL Intermediate Language

All frontends, passes and backends in Yosys operate on a design in RTLIL[1] representation. The only exception are the high-level frontends that use the AST representation as an intermediate step before generating RTLIL data.

In order to avoid reinventing names for the RTLIL classes, they are simply referred to by their full C++ name, i.e. including the `RTLIL::` namespace prefix, in this document.

Figure 4.2 shows a simplified Entity-Relationship Diagram (ER Diagram) of RTLIL. In $1 : N$ relationships the arrow points from the N side to the 1. For example one RTLIL::Design contains N (zero to many) instances of RTLIL::Module. A two-pointed arrow indicates a $1 : 1$ relationship.

The RTLIL::Design is the root object of the RTLIL data structure. There is always one "current design" in memory which passes operate on, frontends add data to and backends convert to exportable formats. But in some cases passes internally generate additional RTLIL::Design objects. For example when a pass is reading an auxiliary Verilog file such as a cell library, it might create an additional RTLIL::Design object and call the Verilog frontend with this other object to parse the cell library.

There is only one active RTLIL::Design object that is used by all frontends, passes and backends called by the user, e.g. using a synthesis script. The RTLIL::Design then contains zero to many RTLIL::Module objects. This corresponds to modules in Verilog or entities in VHDL. Each module in turn contains objects from three different categories:

---

[1] The *Language* in *RTL Intermediate Language* refers to the fact, that RTLIL also has a text representation, usually referred to as *Intermediate Language* (ILANG).
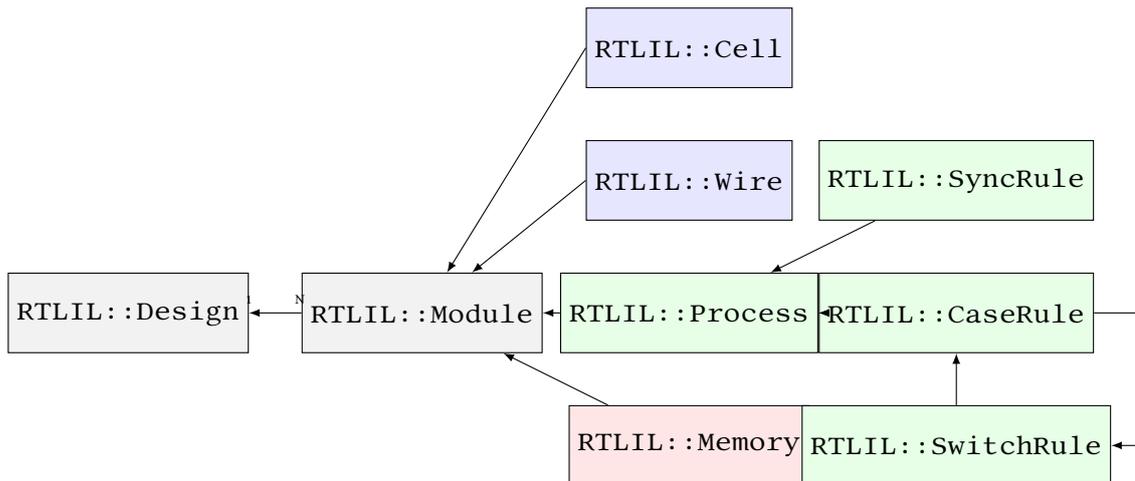
Figure 4.2: Simplified RTLIL Entity-Relationship Diagram

- RTLIL::Cell and RTLIL::Wire objects represent classical netlist data.

- RTLIL::Process objects represent the decision trees (if-then-else statements, etc.) and synchronization declarations (clock signals and sensitivity) from Verilog `always` and VHDL `process` blocks.

- RTLIL::Memory objects represent addressable memories (arrays).

Usually the output of the synthesis procedure is a netlist, i.e. all RTLIL::Process and RTLIL::Memory objects must be replaced by RTLIL::Cell and RTLIL::Wire objects by synthesis passes.

All features of the HDL that cannot be mapped directly to these RTLIL classes must be transformed to an RTLIL-compatible representation by the HDL frontend. This includes Verilog-features such as generate-blocks, loops and parameters.

The following sections contain a more detailed description of the different parts of RTLIL and rationale behind some of the design decisions.

4.2.1 RTLIL Identifiers

All identifiers in RTLIL (such as module names, port names, signal names, cell types, etc.) follow the following naming convention: they must either start with a backslash (\) or a dollar sign ($).

Identifiers starting with a backslash are public visible identifiers. Usually they originate from one of the HDL input files. For example the signal name "`\sig42`" is most likely a signal that was declared using the name "`sig42`" in an HDL input file. On the other hand the signal name "`$sig42`" is an auto-generated signal name. The backends convert all identifiers that start with a dollar sign to identifiers that do not collide with identifiers that start with a backslash.

This has three advantages:

- First, it is impossible that an auto-generated identifier collides with an identifier that was provided by the user.

- Second, the information about which identifiers were originally provided by the user is always available which can help guide some optimizations. For example the "opt_rmunused" tries to preserve signals with a user-provided name but doesn't hesitate to delete signals that have auto-generated names when they just duplicate other signals.

- Third, the delicate job of finding suitable auto-generated public visible names is deferred to one central location. Internally auto-generated names that may hold important information for Yosys developers can be used without disturbing external tools. For example the Verilog backend assigns names in the form *_integer_*.

In order to avoid programming errors, the RTLIL data structures check if all identifiers start with either a backslash or a dollar sign and generate a runtime error if this rule is violated.

All RTLIL identifiers are case sensitive.

4.2.2 RTLIL::Design and RTLIL::Module

The RTLIL::Design object is basically just a container for RTLIL::Module objects. In addition to a list of RTLIL::Module objects the RTLIL::Design also keeps a list of *selected objects*, i.e. the objects that passes should operate on. In most cases the whole design is selected and therefore passes operate on the whole design. But this mechanism can be useful for more complex synthesis jobs in which only parts of the design should be affected by certain passes.

Besides the objects shown in the ER diagram in Fig. 4.2 an RTLIL::Module object contains the following additional properties:

- The module name

- A list of attributes

- A list of connections between wires

- An optional frontend callback used to derive parametrized variations of the module

The attributes can be Verilog attributes imported by the Verilog frontend or attributes assigned by passes. They can be used to store additional metadata about modules or just mark them to be used by certain part of the synthesis script but not by others.

Verilog and VHDL both support parametric modules (known as "generic entities" in VHDL). The RTLIL format does not support parametric modules itself. Instead each module contains a callback function into the AST frontend to generate a parametrized variation of the RTLIL::Module as needed. This callback then returns the auto-generated name of the parametrized variation of the module. (A hash over the parameters and the module name is used to prohibit the same parametrized variation from being generated twice. For modules with only a few parameters, a name directly containing all parameters is generated instead of a hash string.)

4.2.3 RTLIL::Cell and RTLIL::Wire

A module contains zero to many RTLIL::Cell and RTLIL::Wire objects. Objects of these types are used to model netlists. Usually the goal of all synthesis efforts is to convert all modules to a state where the functionality of the module is implemented only by cells from a given cell library and wires to connect these cells with each other. Note that module ports are just wires with a special property.

An RTLIL::Wire object has the following properties:

- The wire name

- A list of attributes

- A width (buses are just wires with a width > 1)

- If the wire is a port: port number and direction (input/output/inout)

As with modules, the attributes can be Verilog attributes imported by the Verilog frontend or attributes assigned by passes.

In Yosys, busses (signal vectors) are represented using a single wire object with a width > 1. So Yosys does not convert signal vectors to individual signals. This makes some aspects of RTLIL more complex but enables Yosys to be used for coarse grain synthesis where the cells of the target architecture operate on entire signal vectors instead of single bit wires.

An RTLIL::Cell object has the following properties:

- The cell name and type

- A list of attributes

- A list of parameters (for parametric cells)

- Cell ports and the connections of ports to wires and constants

The connections of ports to wires are coded by assigning an RTLIL::SigSpec to each cell port. The RTLIL::SigSpec data type is described in the next section.

4.2.4   RTLIL::SigSpec

A "signal" is everything that can be applied to a cell port. I.e.

- Any constant value of arbitrary bit-width
  For example: `1337, 16'b00000010100111001, 1'b1, 1'bx`

- All bits of a wire or a selection of bits from a wire
  For example: `mywire, mywire[24], mywire[15:8]`

- Concatenations of the above
  For example: `{16'd1337, mywire[15:8]}`

The RTLIL::SigSpec data type is used to represent signals. The RTLIL::Cell object contains one RTLIL::SigSpec for each cell port.

In addition, connections between wires are represented using a pair of RTLIL::SigSpec objects. Such pairs are needed in different locations. Therefore the type name RTLIL::SigSig was defined for such a pair.

4.2.5   RTLIL::Process

When a high-level HDL frontend processes behavioural code it splits it up into data path logic (e.g. the expression `a + b` is replaced by the output of an adder that takes `a` and `b` as inputs) and an RTLIL::Process that models the control logic of the behavioural code. Let's consider a simple example:

```verilog
module ff_with_en_and_async_reset(clock, reset, enable, d, q);
input clock, reset, enable, d;
output reg q;
always @(posedge clock, posedge reset)
        if (reset)
                q <= 0;
        else if (enable)
                q <= d;
endmodule
```

In this example there is no data path and therefore the RTLIL::Module generated by the frontend only contains a few RTLIL::Wire objects and an RTLIL::Process. The RTLIL::Process in ILANG syntax:

```
1  process $proc$ff_with_en_and_async_reset.v:4$1
2          assign $0\q[0:0] \q
3          switch \reset
4                  case 1'1
5                          assign $0\q[0:0] 1'0
6                  case
7                          switch \enable
8                                  case 1'1
9                                          assign $0\q[0:0] \d
10                                 case
11                         end
12         end
13         sync posedge \clock
14                 update \q $0\q[0:0]
15         sync posedge \reset
16                 update \q $0\q[0:0]
17 end
```

This RTLIL::Process contains two RTLIL::SyncRule objects, two RTLIL::SwitchRule objects and five RTLIL::CaseRule objects. The wire $0\q[0:0] is an automatically created wire that holds the next value of \q. The lines $2\ldots12$ describe how $0\q[0:0] should be calculated. The lines $13\ldots16$ describe how the value of $0\q[0:0] is used to update \q.

An RTLIL::Process is a container for zero or more RTLIL::SyncRule objects and exactly one RTLIL::CaseRule object, which is called the *root case*.

An RTLIL::SyncRule object contains an (optional) synchronization condition (signal and edge-type) and zero or more assignments (RTLIL::SigSig).

An RTLIL::CaseRule is a container for zero or more assignments (RTLIL::SigSig) and zero or more RTLIL::SwitchRule objects. An RTLIL::SwitchRule objects is a container for zero or more RTLIL::CaseRule objects.

In the above example the lines $2\ldots12$ are the root case. Here $0\q[0:0] is first assigned the old value \q as default value (line 2). The root case also contains an RTLIL::SwitchRule object (lines $3\ldots12$). Such an object is very similar to the C switch statement as it uses a control signal (\reset in this case) to determine which of its cases should be active. The RTLIL::SwitchRule object then contains one RTLIL::CaseRule object per case. In this example there is a case[2] for \reset == 1 that causes $0\q[0:0] to be set (lines 4 and 5) and a default case that in turn contains a switch that sets $0\q[0:0] to the value of \d if \enable is active (lines $6\ldots11$).

The lines $13\ldots16$ then cause \q to be updated whenever there is a positive clock edge on \clock or \reset.

In order to generate such a representation, the language frontend must be able to handle blocking and nonblocking assignments correctly. However, the language frontend does not need to identify the correct type of storage element for the output signal or generate multiplexers for the decision tree. This is done by passes that work on the RTLIL representation. Therefore it is relatively easy to substitute these steps with other algorithms that target different target architectures or perform optimizations or other transformations on the decision trees before further processing them.

One of the first actions performed on a design in RTLIL representation in most synthesis scripts is identifying asynchronous resets. This is usually done using the proc_arst pass. This pass transforms the above example to the following RTLIL::Process:

---

[2]The syntax 1'1 in the ILANG code specifies a constant with a length of one bit (the first "1"), and this bit is a one (the second "1").

```
1  process $proc$ff_with_en_and_async_reset.v:4$1
2         assign $0\q[0:0] \q
3         switch \enable
4                 case 1'1
5                         assign $0\q[0:0] \d
6                 case
7         end
8         sync posedge \clock
9                 update \q $0\q[0:0]
10        sync high \reset
11                update \q 1'0
12 end
```

This pass has transformed the outer RTLIL::SwitchRule into a modified RTLIL::SyncRule object for the \reset signal. Further processing converts the RTLIL::Process into e.g. a d-type flip-flop with asynchronous reset and a multiplexer for the enable signal:

```
1  cell $adff $procdff$6
2          parameter \ARST_POLARITY 1'1
3          parameter \ARST_VALUE 1'0
4          parameter \CLK_POLARITY 1'1
5          parameter \WIDTH 1
6          connect \ARST \reset
7          connect \CLK \clock
8          connect \D $0\q[0:0]
9          connect \Q \q
10 end
11 cell $mux $procmux$3
12         parameter \WIDTH 1
13         connect \A \q
14         connect \B \d
15         connect \S \enable
16         connect \Y $0\q[0:0]
17 end
```

Different combinations of passes may yield different results. Note that $adff and $mux are internal cell types that still need to be mapped to cell types from the target cell library.

Some passes refuse to operate on modules that still contain RTLIL::Process objects as the presence of these objects in a module increases the complexity. Therefore the passes to translate processes to a netlist of cells are usually called early in a synthesis script. The proc pass calls a series of other passes that together perform this conversion in a way that is suitable for most synthesis tasks.

4.2.6 RTLIL::Memory

For every array (memory) in the HDL code an RTLIL::Memory object is created. A memory object has the following properties:

- The memory name

- A list of attributes

- The width of an addressable word

- The size of the memory in number of words

All read accesses to the memory are transformed to $memrd cells and all write accesses to $memwr cells by the language frontend. These cells consist of independent read- and write-ports to the memory. The \MEMID parameter on these cells is used to link them together and to the RTLIL::Memory object they belong to.

The rationale behind using separate cells for the individual ports versus creating a large multiport memory cell right in the language frontend is that the separate $memrd and $memwr cells can be consolidated using resource sharing. As resource sharing is a non-trivial optimization problem where different synthesis tasks can have different requirements it lends itself to do the optimisation in separate passes and merge the RTLIL::Memory objects and $memrd and $memwr cells to multiport memory blocks after resource sharing is completed.

The memory pass performs this conversion and can (depending on the options passed to it) transform the memories directly to d-type flip-flops and address logic or yield multiport memory blocks (represented using $mem cells).

See Sec. 5.1.5 for details about the memory cell types.

4.3 Command Interface and Synthesis Scripts

Yosys reads and processes commands from synthesis scripts, command line arguments and an interactive command prompt. Yosys commands consist of a command name and an optional whitespace separated list of arguments. Commands are terminated using the newline character or a semicolon (;). Empty lines and lines starting with the hash sign (#) are ignored. See Sec. 3.3 for an example synthesis script.

The command help can be used to access the command reference manual.

Most commands can operate not only on the entire design but also specifically on *selected* parts of the design. For example the command dump will print all selected objects in the current design while dump foobar will only print the module foobar and dump * will print the entire design regardless of the current selection.

The selection mechanism is very powerful. For example the command dump */t:$add %x:+[A] */w:* %i will print all wires that are connected to the \A port of a $add cell. Detailed documentation of the select framework can be found in the command reference for the select command.

4.4 Source Tree and Build System

The Yosys source tree is organized into the following top-level directories:

- backends/
  This directory contains a subdirectory for each of the backend modules.

- frontends/
  This directory contains a subdirectory for each of the frontend modules.

- kernel/
  This directory contains all the core functionality of Yosys. This includes the functions and definitions for working with the RTLIL data structures (rtlil.h and rtlil.cc), the main() function (driver.cc), the internal framework for generating log messages (log.h and log.cc), the internal framework for registering and calling passes (register.h and register.cc), some core commands that are not really passes (select.cc, show.cc, ...) and a couple of other small utility libraries.

- passes/
  This directory contains a subdirectory for each pass or group of passes. For example as of this writing the directory passes/opt/ contains the code for seven passes: opt, opt_expr, opt_muxtree, opt_reduce, opt_rmdff, opt_rmunused and opt_merge.

- `techlibs/`
  This directory contains simulation models and standard implementations for the cells from the internal cell library.

- `tests/`
  This directory contains a couple of test cases. Most of the smaller tests are executed automatically when `make test` is called. The larger tests must be executed manually. Most of the larger tests require downloading external HDL source code and/or external tools. The tests range from comparing simulation results of the synthesized design to the original sources to logic equivalence checking of entire CPU cores.

The top-level Makefile includes `frontends/*/Makefile.inc`, `passes/*/Makefile.inc` and `backends/*/Makefile.inc`. So when extending Yosys it is enough to create a new directory in `frontends/`, `passes/` or `backends/` with your sources and a `Makefile.inc`. The Yosys kernel automatically detects all commands linked with Yosys. So it is not needed to add additional commands to a central list of commands.

Good starting points for reading example source code to learn how to write passes are `passes/opt/opt_rmdff.cc` and `passes/opt/opt_merge.cc`.

See the top-level README file for a quick *Getting Started* guide and build instructions. The Yosys build is based solely on Makefiles.

Users of the Qt Creator IDE can generate a QT Creator project file using `make qtcreator`. Users of the Eclipse IDE can use the "Makefile Project with Existing Code" project type in the Eclipse "New Project" dialog (only available after the CDT plugin has been installed) to create an Eclipse project in order to programming extensions to Yosys or just browse the Yosys code base.

Chapter 5

Internal Cell Library

Most of the passes in Yosys operate on netlists, i.e. they only care about the RTLIL::Wire and RTLIL::Cell objects in an RTLIL::Module. This chapter discusses the cell types used by Yosys to represent a behavioural design internally.

This chapter is split in two parts. In the first part the internal RTL cells are covered. These cells are used to represent the design on a coarse grain level. Like in the original HDL code on this level the cells operate on vectors of signals and complex cells like adders exist. In the second part the internal gate cells are covered. These cells are used to represent the design on a fine-grain gate-level. All cells from this category operate on single bit signals.

5.1 RTL Cells

Most of the RTL cells closely resemble the operators available in HDLs such as Verilog or VHDL. Therefore Verilog operators are used in the following sections to define the behaviour of the RTL cells.

Note that all RTL cells have parameters indicating the size of inputs and outputs. When passes modify RTL cells they must always keep the values of these parameters in sync with the size of the signals connected to the inputs and outputs.

Simulation models for the RTL cells can be found in the file `techlibs/common/simlib.v` in the Yosys source tree.

5.1.1 Unary Operators

All unary RTL cells have one input port `\A` and one output port `\Y`. They also have the following parameters:

- `\A_SIGNED`
  Set to a non-zero value if the input `\A` is signed and therefore should be sign-extended when needed.

- `\A_WIDTH`
  The width of the input port `\A`.

- `\Y_WIDTH`
  The width of the output port `\Y`.

Table 5.1 lists all cells for unary RTL operators.

Note that `$reduce_or` and `$reduce_bool` actually represent the same logic function. But the HDL frontends generate them in different situations. A `$reduce_or` cell is generated when the prefix `|` operator is being used. A `$reduce_bool` cell is generated when a bit vector is used as a condition in an `if`-statement or `?:`-expression.

Verilog	Cell Type
Y = ~A	$not
Y = +A	$pos
Y = -A	$neg
Y = &A	$reduce_and
Y = |A	$reduce_or
Y = ^A	$reduce_xor
Y = ~^A	$reduce_xnor
Y = |A	$reduce_bool
Y = !A	$logic_not

Table 5.1: Cell types for unary operators with their corresponding Verilog expressions.

5.1.2 Binary Operators

All binary RTL cells have two input ports \A and \B and one output port \Y. They also have the following parameters:

- \A_SIGNED
  Set to a non-zero value if the input \A is signed and therefore should be sign-extended when needed.

- \A_WIDTH
  The width of the input port \A.

- \B_SIGNED
  Set to a non-zero value if the input \B is signed and therefore should be sign-extended when needed.

- \B_WIDTH
  The width of the input port \B.

- \Y_WIDTH
  The width of the output port \Y.

Table 5.2 lists all cells for binary RTL operators.

5.1.3 Multiplexers

Multiplexers are generated by the Verilog HDL frontend for ?:-expressions. Multiplexers are also generated by the proc pass to map the decision trees from RTLIL::Process objects to logic.

The simplest multiplexer cell type is $mux. Cells of this type have a \WIDTH parameter and data inputs \A and \B and a data output \Y, all of the specified width. This cell also has a single bit control input \S. If \S is 0 the value from the \A input is sent to the output, if it is 1 the value from the \B input is sent to the output. So the $mux cell implements the function Y = S ? B : A.

The $pmux cell is used to multiplex between many inputs using a one-hot select signal. Cells of this type have a \WIDTH and a \S_WIDTH parameter and inputs \A, \B, and \S and an output \Y. The \S input is \S_WIDTH bits wide. The \A input and the output are both \WIDTH bits wide and the \B input is \WIDTH*\S_WIDTH bits wide. When all bits of \S are zero, the value from \A input is sent to the output. If the n'th bit from \S is set, the value n'th \WIDTH bits wide slice of the \B input is sent to the output. When more than one bit from \S is set the output is undefined. Cells of this type are used to model "parallel cases" (defined by using the parallel_case attribute or detected by an optimization).

Behavioural code with cascaded if-then-else- and case-statements usually results in trees of multiplexer cells. Many passes (from various optimizations to FSM extraction) heavily depend on these multiplexer trees to understand dependencies between signals. Therefore optimizations should not break these multiplexer trees (e.g. by replacing a multiplexer between a calculated signal and a constant zero with an $and gate).

Verilog	Cell Type		Verilog	Cell Type
Y = A & B	$and		Y = A < B	$lt
Y = A | B	$or		Y = A <= B	$le
Y = A ^ B	$xor		Y = A == B	$eq
Y = A ~^ B	$xnor		Y = A != B	$ne
Y = A << B	$shl		Y = A >= B	$ge
Y = A >> B	$shr		Y = A > B	$gt
Y = A <<< B	$sshl		Y = A + B	$add
Y = A >>> B	$sshr		Y = A - B	$sub
Y = A && B	$logic_and		Y = A * B	$mul
Y = A || B	$logic_or		Y = A / B	$div
Y = A === B	$eqx		Y = A % B	$mod
Y = A !== B	$nex		Y = A ** B	$pow

Table 5.2: Cell types for binary operators with their corresponding Verilog expressions.

5.1.4 Registers

D-Type Flip-Flops are represented by $dff cells. These cells have a clock port \CLK, an input port \D and an output port \Q. The following parameters are available for $dff cells:

- \WIDTH
  The width of input \D and output \Q.

- \CLK_POLARITY
  Clock is active on the positive edge if this parameter has the value 1'b1 and on the negative edge if this parameter is 1'b0.

D-Type Flip-Flops with asynchronous resets are represented by $adff cells. As the $dff cells they have \CLK, \D and \Q ports. In addition they also have a single-bit \ARST input port for the reset pin and the following additional two parameters:

- \ARST_POLARITY
  The asynchronous reset is high-active if this parameter has the value 1'b1 and low-active if this parameter is 1'b0.

- \ARST_VALUE
  The state of \Q will be set to this value when the reset is active.

Note that the $adff cell can only be used when the reset value is constant.

Usually these cells are generated by the proc pass using the information in the designs RTLIL::Process objects.

FIXME:
Add information about $sr cells (set-reset flip-flops) and d-type latches.

5.1.5 Memories

Memories are either represented using RTLIL::Memory objects and $memrd and $memwr cells or simply by using $mem cells.

In the first alternative the RTLIL::Memory objects hold the general metadata for the memory (bit width, size in number of words, etc.) and for each port a $memrd (read port) or $memwr (write port) cell is created. Having

individual cells for read and write ports has the advantage that they can be consolidated using resource sharing passes. In some cases this drastically reduces the number of required ports on the memory cell.

The $memrd cells have a clock input \CLK, an enable input \EN, an address input \ADDR, and a data output \DATA. They also have the following parameters:

- \MEMID
  The name of the RTLIL::Memory object that is associated with this read port.

- \ABITS
  The number of address bits (width of the \ADDR input port).

- \WIDTH
  The number of data bits (width of the \DATA output port).

- \CLK_ENABLE
  When this parameter is non-zero, the clock is used. Otherwise this read port is asynchronous and the \CLK input is not used.

- \CLK_POLARITY
  Clock is active on the positive edge if this parameter has the value 1'b1 and on the negative edge if this parameter is 1'b0.

- \TRANSPARENT
  If this parameter is set to 1'b1, a read and write to the same address in the same cycle will return the new value. Otherwise the old value is returned.

The $memwr cells have a clock input \CLK, an enable input \EN (one enable bit for each data bit), an address input \ADDR and a data input \DATA. They also have the following parameters:

- \MEMID
  The name of the RTLIL::Memory object that is associated with this read port.

- \ABITS
  The number of address bits (width of the \ADDR input port).

- \WIDTH
  The number of data bits (width of the \DATA output port).

- \CLK_ENABLE
  When this parameter is non-zero, the clock is used. Otherwise this read port is asynchronous and the \CLK input is not used.

- \CLK_POLARITY
  Clock is active on positive edge if this parameter has the value 1'b1 and on the negative edge if this parameter is 1'b0.

- \PRIORITY
  The cell with the higher integer value in this parameter wins a write conflict.

The HDL frontend models a memory using RTLIL::Memory objects and asynchronous $memrd and $memwr cells. The memory pass (i.e. its various sub-passes) migrates $dff cells into the $memrd and $memwr cells making them synchronous, then converts them to a single $mem cell and (optionally) maps this cell type to $dff cells for the individual words and multiplexer-based address decoders for the read and write interfaces. When the last step is disabled or not possible, a $mem cell is left in the design.

The $mem cell provides the following parameters:

- \MEMID
  The name of the original RTLIL::Memory object that became this $mem cell.

- \SIZE
  The number of words in the memory.

- \ABITS
  The number of address bits.

- \WIDTH
  The number of data bits per word.

- \RD_PORTS
  The number of read ports on this memory cell.

- \RD_CLK_ENABLE
  This parameter is \RD_PORTS bits wide, containing a clock enable bit for each read port.

- \RD_CLK_POLARITY
  This parameter is \RD_PORTS bits wide, containing a clock polarity bit for each read port.

- \RD_TRANSPARENT
  This parameter is \RD_PORTS bits wide, containing a transparent bit for each read port.

- \WR_PORTS
  The number of write ports on this memory cell.

- \WR_CLK_ENABLE
  This parameter is \WR_PORTS bits wide, containing a clock enable bit for each write port.

- \WR_CLK_POLARITY
  This parameter is \WR_PORTS bits wide, containing a clock polarity bit for each write port.

The $mem cell has the following ports:

- \RD_CLK
  This input is \RD_PORTS bits wide, containing all clock signals for the read ports.

- \RD_EN
  This input is \RD_PORTS bits wide, containing all enable signals for the read ports.

- \RD_ADDR
  This input is \RD_PORTS*\ABITS bits wide, containing all address signals for the read ports.

- \RD_DATA
  This input is \RD_PORTS*\WIDTH bits wide, containing all data signals for the read ports.

- \WR_CLK
  This input is \WR_PORTS bits wide, containing all clock signals for the write ports.

- \WR_EN
  This input is \WR_PORTS*\WIDTH bits wide, containing all enable signals for the write ports.

- \WR_ADDR
  This input is \WR_PORTS*\ABITS bits wide, containing all address signals for the write ports.

- \WR_DATA
  This input is \WR_PORTS*\WIDTH bits wide, containing all data signals for the write ports.

The techmap pass can be used to manually map $mem cells to specialized memory cells on the target architecture, such as block ram resources on an FPGA.

Verilog	Cell Type
Y = ~A	$_NOT_
Y = A & B	$_AND_
Y = A | B	$_OR_
Y = A ^ B	$_XOR_
Y = S ? B : A	$_MUX_
always @(**negedge** C) Q <= D	$_DFF_N_
always @(**posedge** C) Q <= D	$_DFF_P_

ClkEdge	RstLvl	RstVal	Cell Type
negedge	0	0	$_DFF_NN0_
negedge	0	1	$_DFF_NN1_
negedge	1	0	$_DFF_NP0_
negedge	1	1	$_DFF_NP1_
posedge	0	0	$_DFF_PN0_
posedge	0	1	$_DFF_PN1_
posedge	1	0	$_DFF_PP0_
posedge	1	1	$_DFF_PP1_

Table 5.3: Cell types for gate level logic networks

5.1.6 Finite State Machines

FIXME:

Add a brief description of the $fsm cell type.

5.2 Gates

For gate level logic networks, fixed function single bit cells are used that do not provide any parameters.

Simulation models for these cells can be found in the file techlibs/common/simcells.v in the Yosys source tree.

Table 5.3 lists all cell types used for gate level logic. The cell types $_NOT_, $_AND_, $_OR_, $_XOR_ and $_MUX_ are used to model combinatorial logic. The cell types $_DFF_N_ and $_DFF_P_ represent d-type flip-flops.

The cell types $_DFF_NN0_, $_DFF_NN1_, $_DFF_NP0_, $_DFF_NP1_, $_DFF_PN0_, $_DFF_PN1_, $_DFF_PP0_ and $_DFF_PP1_ implement d-type flip-flops with asynchronous resets. The values in the table for these cell types relate to the following Verilog code template, where $RstEdge$ is **posedge** if $RstLvl$ if 1, and **negedge** otherwise.

```
always @(ClkEdge C, RstEdge R)
        if (R == RstLvl)
                Q <= RstVal;
        else
                Q <= D;
```

In most cases gate level logic networks are created from RTL networks using the techmap pass. The flip-flop cells from the gate level logic network can be mapped to physical flip-flop cells from a Liberty file using the dfflibmap pass. The combinatorial logic cells can be mapped to physical cells from a Liberty file via ABC [27] using the abc pass.

FIXME:

Add information about $assert, $assume, $equiv, $initstate, $anyconst, and $anyseq cells.

FIXME:

Add information about `$slice` and `$concat` cells.

FIXME:

Add information about `$lut` and `$sop` cells.

FIXME:

Add information about `$alu`, `$macc`, `$fa`, and `$lcu` cells.

FIXME:

Add information about `$ff` and `$_FF_` cells.

FIXME:

Add information about `$dffe`, `$dffsr`, `$dlatch`, and `$dlatchsr` cells.

FIXME:

Add information about $_DFFE_??_, $_DFFSR_???_, $_DLATCH_?_, and $_DLATCHSR_???_ cells.

FIXME:

Add information about $_NAND_, $_NOR_, $_XNOR_, $_AOI3_, $_OAI3_, $_AOI4_, and $_OAI4_ cells.

Chapter 6

Programming Yosys Extensions

This chapter contains some bits and pieces of information about programming yosys extensions. Also consult the section on programming in the "Yosys Presentation" (can be downloaded from the Yosys website as PDF) and don't be afraid to ask questions on the Yosys Subreddit.

6.1 The "CodingReadme" File

The following is an excerpt of the CodingReadme file from the Yosys source tree.

CodingReadme

```
 8  Getting Started
 9  ===============
10
11
12  Outline of a Yosys command
13  --------------------------
14
15  Here is a the C++ code for a "hello_world" Yosys command (hello.cc):
16
17          #include "kernel/yosys.h"
18
19          USING_YOSYS_NAMESPACE
20          PRIVATE_NAMESPACE_BEGIN
21
22          struct HelloWorldPass : public Pass {
23                  HelloWorldPass() : Pass("hello_world") { }
24                  virtual void execute(vector<string>, Design*) {
25                          log("Hello World!\n");
26                  }
27          } HelloWorldPass;
28
29          PRIVATE_NAMESPACE_END
30
31  This can be built into a Yosys module using the following command:
32
33          yosys-config --exec --cxx --cxxflags --ldflags -o hello.so -shared hello.cc --
34
```

```
35  Or short:
36
37          yosys-config --build hello.so hello.cc
38
39  And then executed using the following command:
40
41          yosys -m hello.so -p hello_world
42
43
44  Yosys Data Structures
45  ---------------------
46
47  Here is a short list of data structures that you should make yourself familiar
48  with before you write C++ code for Yosys. The following data structures are all
49  defined when "kernel/yosys.h" is included and USING_YOSYS_NAMESPACE is used.
50
51    1. Yosys Container Classes
52
53  Yosys uses dict<K, T> and pool<T> as main container classes. dict<K, T> is
54  essentially a replacement for std::unordered_map<K, T> and pool<T> is a
55  replacement for std::unordered_set<T>. The main characteristics are:
56
57          - dict<K, T> and pool<T> are about 2x faster than the std containers
58
59          - references to elements in a dict<K, T> or pool<T> are invalidated by
60            insert and remove operations (similar to std::vector<T> on push_back()).
61
62          - some iterators are invalidated by erase(). specifically, iterators
63            that have not passed the erased element yet are invalidated. (erase()
64            itself returns valid iterator to the next element.)
65
66          - no iterators are invalidated by insert(). elements are inserted at
67            begin(). i.e. only a new iterator that starts at begin() will see the
68            inserted elements.
69
70          - the method .count(key, iterator) is like .count(key) but only
71            considers elements that can be reached via the iterator.
72
73          - iterators can be compared. it1 < it2 means that the position of t2
74            can be reached via t1 but not vice versa.
75
76          - the method .sort() can be used to sort the elements in the container
77            the container stays sorted until elements are added or removed.
78
79          - dict<K, T> and pool<T> will have the same order of iteration across
80            all compilers, standard libraries and architectures.
81
82  In addition to dict<K, T> and pool<T> there is also an idict<K> that
83  creates a bijective map from K to the integers. For example:
84
85          idict<string, 42> si;
86          log("%d\n", si("hello"));      // will print 42
87          log("%d\n", si("world"));      // will print 43
88          log("%d\n", si.at("world"));   // will print 43
```

```
89          log("%d\n", si.at("dummy"));   // will throw exception
90          log("%s\n", si[42].c_str()));  // will print hello
91          log("%s\n", si[43].c_str()));  // will print world
92          log("%s\n", si[44].c_str()));  // will throw exception
```

It is not possible to remove elements from an idict.

Finally mfp<K> implements a merge-find set data structure (aka. disjoint-set or union-find) over the type K ("mfp" = merge-find-promote).

   2. Standard STL data types

In Yosys we use std::vector<T> and std::string whenever applicable. When dict<K, T> and pool<T> are not suitable then std::map<K, T> and std::set<T> are used instead.

The types std::vector<T> and std::string are also available as vector<T> and string in the Yosys namespace.

   3. RTLIL objects

The current design (essentially a collection of modules, each defined by a netlist) is stored in memory using RTLIL object (declared in kernel/rtlil.h, automatically included by kernel/yosys.h). You should glance over at least the declarations for the following types in kernel/rtlil.h:

```
        RTLIL::IdString
                This is a handle for an identifier (e.g. cell or wire name).
                It feels a lot like a std::string, but is only a single int
                in size. (The actual string is stored in a global lookup
                table.)

        RTLIL::SigBit
                A single signal bit. I.e. either a constant state (0, 1,
                x, z) or a single bit from a wire.

        RTLIL::SigSpec
                Essentially a vector of SigBits.

        RTLIL::Wire
        RTLIL::Cell
                The building blocks of the netlist in a module.

        RTLIL::Module
        RTLIL::Design
                The module is a container with connected cells and wires
                in it. The design is a container with modules in it.
```

All this types are also available without the RTLIL:: prefix in the Yosys namespace.

   4. SigMap and other Helper Classes

There are a couple of additional helper classes that are in wide use

143 | in Yosys. Most importantly there is SigMap (declared in kernel/sigtools.h).
144 |
145 | When a design has many wires in it that are connected to each other, then a
146 | single signal bit can have multiple valid names. The SigMap object can be used
147 | to map SigSpecs or SigBits to unique SigSpecs and SigBits that consistently
148 | only use one wire from such a group of connected wires. For example:
149 |
150 | SigBit a = module->addWire(NEW_ID);
151 | SigBit b = module->addWire(NEW_ID);
152 | module->connect(a, b);
153 |
154 | log("%d\n", a == b); // will print 0
155 |
156 | SigMap sigmap(module);
157 | log("%d\n", sigmap(a) == sigmap(b)); // will print 1
158 |
159 |
160 | Using the RTLIL Netlist Format
161 | ------------------------------
162 |
163 | In the RTLIL netlist format the cell ports contain SigSpecs that point to the
164 | Wires. There are no references in the other direction. This has two direct
165 | consequences:
166 |
167 | (1) It is very easy to go from cells to wires but hard to go in the other way.
168 |
169 | (2) There is no danger in removing cells from the netlists, but removing wires
170 | can break the netlist format when there are still references to the wire
171 | somewhere in the netlist.
172 |
173 | The solution to (1) is easy: Create custom indexes that allow you to make fast
174 | lookups for the wire-to-cell direction. You can either use existing generic
175 | index structures to do that (such as the ModIndex class) or write your own
176 | index. For many application it is simplest to construct a custom index. For
177 | example:
178 |
179 | SigMap sigmap(module);
180 | dict<SigBit, Cell*> sigbit_to_driver_index;
181 |
182 | for (auto cell : module->cells())
183 | for (auto &conn : cell->connections())
184 | if (cell->output(conn.first))
185 | for (auto bit : sigmap(conn.second))
186 | sigbit_to_driver_index[bit] = cell;
187 |
188 | Regarding (2): There is a general theme in Yosys that you don't remove wires
189 | from the design. You can rename them, unconnect them, but you do not actually remove
190 | the Wire object from the module. Instead you let the "clean" command take care
191 | of the dangling wires. On the other hand it is safe to remove cells (as long as
192 | you make sure this does not invalidate a custom index you are using in your code).
193 |
194 |
195 | Example Code
196 | ------------

```
The following yosys commands are a good starting point if you are looking for examples
of how to use the Yosys API:

        manual/CHAPTER_Prog/stubnets.cc
        manual/PRESENTATION_Prog/my_cmd.cc


Notes on the existing codebase
------------------------------

For historical reasons not all parts of Yosys adhere to the current coding
style. When adding code to existing parts of the system, adhere to this guide
for the new code instead of trying to mimic the style of the surrounding code.



Coding Style
============


Formatting of code
------------------

- Yosys code is using tabs for indentation. Tabs are 8 characters.

- A continuation of a statement in the following line is indented by
  two additional tabs.

- Lines are as long as you want them to be. A good rule of thumb is
  to break lines at about column 150.

- Opening braces can be put on the same or next line as the statement
  opening the block (if, switch, for, while, do). Put the opening brace
  on its own line for larger blocks, especially blocks that contains
  blank lines.

- Otherwise stick to the Linux Kernel Coding Style:
      https://www.kernel.org/doc/Documentation/CodingStyle


C++ Language
------------

Yosys is written in C++11. At the moment only constructs supported by
gcc 4.8 are allowed in Yosys code. This will change in future releases.

In general Yosys uses "int" instead of "size_t". To avoid compiler
warnings for implicit type casts, always use "GetSize(foobar)" instead
of "foobar.size()". (GetSize() is defined in kernel/yosys.h)

Use range-based for loops whenever applicable.
```

6.2 The "stubsnets" Example Module

The following is the complete code of the "stubsnets" example module. It is included in the Yosys source distribution as `manual/CHAPTER_Prog/stubnets.cc`.

stubnets.cc

```
1   // This is free and unencumbered software released into the public domain.
2   //
3   // Anyone is free to copy, modify, publish, use, compile, sell, or
4   // distribute this software, either in source code form or as a compiled
5   // binary, for any purpose, commercial or non-commercial, and by any
6   // means.
7
8   #include "kernel/yosys.h"
9   #include "kernel/sigtools.h"
10
11  #include <string>
12  #include <map>
13  #include <set>
14
15  USING_YOSYS_NAMESPACE
16  PRIVATE_NAMESPACE_BEGIN
17
18  // this function is called for each module in the design
19  static void find_stub_nets(RTLIL::Design *design, RTLIL::Module *module, bool report_b
20  {
21          // use a SigMap to convert nets to a unique representation
22          SigMap sigmap(module);
23
24          // count how many times a single-bit signal is used
25          std::map<RTLIL::SigBit, int> bit_usage_count;
26
27          // count output lines for this module (needed only for summary output at the e
28          int line_count = 0;
29
30          log("Looking_for_stub_wires_in_module_%s:\n", RTLIL::id2cstr(module->name));
31
32          // For all ports on all cells
33          for (auto &cell_iter : module->cells_)
34          for (auto &conn : cell_iter.second->connections())
35          {
36                  // Get the signals on the port
37                  // (use sigmap to get a uniqe signal name)
38                  RTLIL::SigSpec sig = sigmap(conn.second);
39
40                  // add each bit to bit_usage_count, unless it is a constant
41                  for (auto &bit : sig)
42                          if (bit.wire != NULL)
43                                  bit_usage_count[bit]++;
44          }
45
46          // for each wire in the module
47          for (auto &wire_iter : module->wires_)
48          {
```

```
49                           RTLIL::Wire *wire = wire_iter.second;
50
51                           // .. but only selected wires
52                           if (!design->selected(module, wire))
53                                   continue;
54
55                           // add +1 usage if this wire actually is a port
56                           int usage_offset = wire->port_id > 0 ? 1 : 0;
57
58                           // we will record which bits of the (possibly multi-bit) wire are stub
59                           std::set<int> stub_bits;
60
61                           // get a signal description for this wire and split it into separate b
62                           RTLIL::SigSpec sig = sigmap(wire);
63
64                           // for each bit (unless it is a constant):
65                           // check if it is used at least two times and add to stub_bits otherwi
66                           for (int i = 0; i < GetSize(sig); i++)
67                                   if (sig[i].wire != NULL && (bit_usage_count[sig[i]] + usage_of
68                                           stub_bits.insert(i);
69
70                           // continue if no stub bits found
71                           if (stub_bits.size() == 0)
72                                   continue;
73
74                           // report stub bits and/or stub wires, don't report single bits
75                           // if called with report_bits set to false.
76                           if (GetSize(stub_bits) == GetSize(sig)) {
77                                   log("  found stub wire: %s\n", RTLIL::id2cstr(wire->name));
78                           } else {
79                                   if (!report_bits)
80                                           continue;
81                                   log("  found wire with stub bits: %s [", RTLIL::id2cstr(wire->
82                                   for (int bit : stub_bits)
83                                           log("%s%d", bit == *stub_bits.begin() ? "" : ", ", bit
84                                   log("]\n");
85                           }
86
87                           // we have outputted a line, increment summary counter
88                           line_count++;
89                   }
90
91           // report summary
92           if (report_bits)
93                   log("  found %d stub wires or wires with stub bits.\n", line_count);
94           else
95                   log("  found %d stub wires.\n", line_count);
96 }
97
98 // each pass contains a singleton object that is derived from Pass
99 struct StubnetsPass : public Pass {
100        StubnetsPass() : Pass("stubnets") { }
101        virtual void execute(std::vector<std::string> args, RTLIL::Design *design)
102        {
```

```
103                     // variables to mirror information from passed options
104                     bool report_bits = 0;
105
106                     log_header(design, "Executing STUBNETS pass (find_stub_nets).\n");
107
108                     // parse options
109                     size_t argidx;
110                     for (argidx = 1; argidx < args.size(); argidx++) {
111                             std::string arg = args[argidx];
112                             if (arg == "-report_bits") {
113                                     report_bits = true;
114                                     continue;
115                             }
116                             break;
117                     }
118
119                     // handle extra options (e.g. selection)
120                     extra_args(args, argidx, design);
121
122                     // call find_stub_nets() for each module that is either
123                     // selected as a whole or contains selected objects.
124                     for (auto &it : design->modules_)
125                             if (design->selected_module(it.first))
126                                     find_stub_nets(design, it.second, report_bits);
127             }
128 } StubnetsPass;
129
130 PRIVATE_NAMESPACE_END
```

Makefile

```
1  test: stubnets.so
2          yosys -ql test1.log -m ./stubnets.so test.v -p "stubnets"
3          yosys -ql test2.log -m ./stubnets.so test.v -p "opt; stubnets"
4          yosys -ql test3.log -m ./stubnets.so test.v -p "techmap; opt; stubnets -report
5          tail test1.log test2.log test3.log
6
7  stubnets.so: stubnets.cc
8          yosys-config --exec --cxx --cxxflags -I../.. --ldflags -o $@ -shared $^ --ldli
9
10 clean:
11         rm -f test1.log test2.log test3.log
12         rm -f stubnets.so stubnets.d
```

test.v

```
1  module uut(in1, in2, in3, out1, out2);
2
3  input [8:0] in1, in2, in3;
4  output [8:0] out1, out2;
5
6  assign out1 = in1 + in2 + (in3 >> 4);
7
8  endmodule
```

Chapter 7

The Verilog and AST Frontends

This chapter provides an overview of the implementation of the Yosys Verilog and AST frontends. The Verilog frontend reads Verilog-2005 code and creates an abstract syntax tree (AST) representation of the input. This AST representation is then passed to the AST frontend that converts it to RTLIL data, as illustrated in Fig. 7.1.

7.1   Transforming Verilog to AST

The *Verilog frontend* converts the Verilog sources to an internal AST representation that closely resembles the structure of the original Verilog code. The Verilog frontend consists of three components, the *Preprocessor*, the *Lexer* and the *Parser*.

The source code to the Verilog frontend can be found in `frontends/verilog/` in the Yosys source tree.
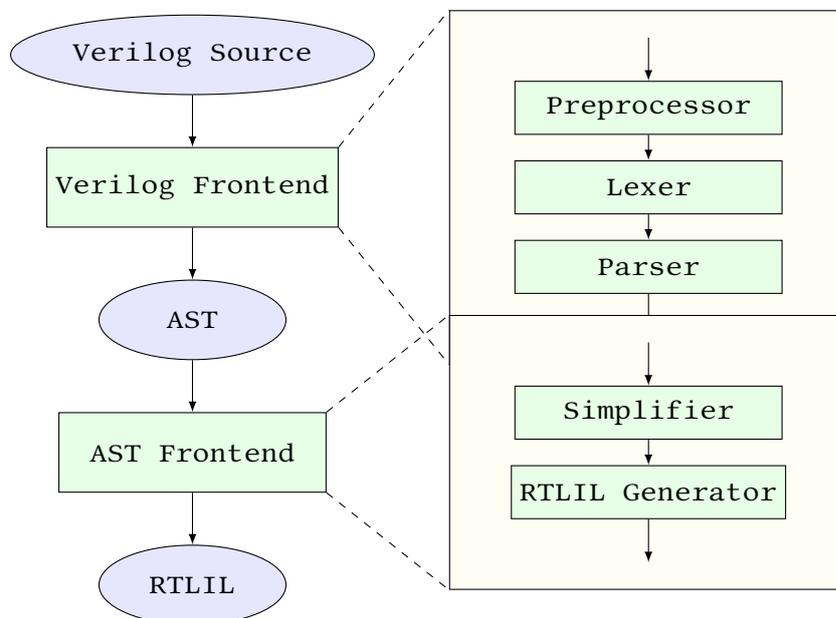
Figure 7.1: Simplified Verilog to RTLIL data flow

7.1.1 The Verilog Preprocessor

The Verilog preprocessor scans over the Verilog source code and interprets some of the Verilog compiler directives such as `'include`, `'define` and `'ifdef`.

It is implemented as a C++ function that is passed a file descriptor as input and returns the pre-processed Verilog code as a `std::string`.

The source code to the Verilog Preprocessor can be found in `frontends/verilog/preproc.cc` in the Yosys source tree.

7.1.2 The Verilog Lexer

The Verilog Lexer is written using the lexer generator *flex* [17]. Its source code can be found in `frontends/verilog/lexer.l` in the Yosys source tree. The lexer does little more than identifying all keywords and literals recognised by the Yosys Verilog frontend.

The lexer keeps track of the current location in the Verilog source code using some global variables. These variables are used by the constructor of AST nodes to annotate each node with the source code location it originated from.

Finally the lexer identifies and handles special comments such as "*//synopsys translate_off*" and "*//synopsys full_case*". (It is recommended to use `'ifdef` constructs instead of the Synsopsys translate_on/off comments and attributes such as (* full_case *) over "*//synopsys full_case*" whenever possible.)

7.1.3 The Verilog Parser

The Verilog Parser is written using the parser generator *bison* [18]. Its source code can be found in `frontends/verilog/parser.y` in the Yosys source tree.

It generates an AST using the `AST::AstNode` data structure defined in `frontends/ast/ast.h`. An `AST::AstNode` object has the following properties:

- **The node type**
  This enum (`AST::AstNodeType`) specifies the role of the node. Table 7.1 contains a list of all node types.

- **The child nodes**
  This is a list of pointers to all children in the abstract syntax tree.

- **Attributes**
  As almost every AST node might have Verilog attributes assigned to it, the `AST::AstNode` has direct support for attributes. Note that the attribute values are again AST nodes.

- **Node content**
  Each node might have additional content data. A series of member variables exist to hold such data. For example the member `std::string str` can hold a string value and is used e.g. in the `AST_IDENTIFIER` node type to store the identifier name.

- **Source code location**
  Each `AST::AstNode` is automatically annotated with the current source code location by the `AST::AstNode` constructor. It is stored in the `std::string filename` and `int linenum` member variables.

The `AST::AstNode` constructor can be called with up to two child nodes that are automatically added to the list of child nodes for the new object. This simplifies the creation of AST nodes for simple expressions a bit. For example the bison code for parsing multiplications:

```
1    basic_expr '*' attr basic_expr {
2            $$ = new AstNode(AST_MUL, $1, $4);
3            append_attr($$, $3);
4    } |
```

The generated AST data structure is then passed directly to the AST frontend that performs the actual conversion to RTLIL.

Note that the Yosys command `read_verilog` provides the options `-yydebug` and `-dump_ast` that can be used to print the parse tree or abstract syntax tree respectively.

7.2   Transforming AST to RTLIL

The *AST Frontend* converts a set of modules in AST representation to modules in RTLIL representation and adds them to the current design. This is done in two steps: *simplification* and *RTLIL generation.*

The source code to the AST frontend can be found in `frontends/ast/` in the Yosys source tree.

7.2.1   AST Simplification

A full-featured AST is too complex to be transformed into RTLIL directly. Therefore it must first be brought into a simpler form. This is done by calling the `AST::AstNode::simplify()` method of all `AST_MODULE` nodes in the AST. This initiates a recursive process that performs the following transformations on the AST data structure:

AST Node Type	Corresponding Verilog Construct
AST_NONE	This Node type should never be used.
AST_DESIGN	This node type is used for the top node of the AST tree. It has no corresponding Verilog construct.
AST_MODULE, AST_TASK, AST_FUNCTION	**module**, **task** and **function**
AST_WIRE	**input**, **output**, **wire**, **reg** and **integer**
AST_MEMORY	Verilog Arrays
AST_AUTOWIRE	Created by the simplifier when an undeclared signal name is used.
AST_PARAMETER, AST_LOCALPARAM	**parameter** and **localparam**
AST_PARASET	Parameter set in cell instantiation
AST_ARGUMENT	Port connection in cell instantiation
AST_RANGE	Bit-Index in a signal or element index in array
AST_CONSTANT	A literal value
AST_CELLTYPE	The type of cell in cell instantiation
AST_IDENTIFIER	An Identifier (signal name in expression or cell/task/etc. name in other contexts)
AST_PREFIX	Construct an identifier in the form `<prefix>[<index>].<suffix>` (used only in advanced generate constructs)
AST_FCALL, AST_TCALL	Call to function or task
AST_TO_SIGNED, AST_TO_UNSIGNED	The `$signed()` and `$unsigned()` functions

Table 7.1: AST node types with their corresponding Verilog constructs.
(continued on next page)

54

AST Node Type	Corresponding Verilog Construct
AST_CONCAT AST_REPLICATE	The {...} and {...{...}} operators
AST_BIT_NOT, AST_BIT_AND, AST_BIT_OR, AST_BIT_XOR, AST_BIT_XNOR	The bitwise operators ~, &, |, ^ and ~^
AST_REDUCE_AND, AST_REDUCE_OR, AST_REDUCE_XOR, AST_REDUCE_XNOR	The unary reduction operators ~, &, |, ^ and ~^
AST_REDUCE_BOOL	Conversion from multi-bit value to boolean value (equivalent to AST_REDUCE_OR)
AST_SHIFT_LEFT, AST_SHIFT_RIGHT, AST_SHIFT_SLEFT, AST_SHIFT_SRIGHT	The shift operators <<, >>, <<< and >>>
AST_LT, AST_LE, AST_EQ, AST_NE, AST_GE, AST_GT	The relational operators <, <=, ==, !=, >= and >
AST_ADD, AST_SUB, AST_MUL, AST_DIV, AST_MOD, AST_POW	The binary operators +, -, *, /, % and **
AST_POS, AST_NEG	The prefix operators + and -
AST_LOGIC_AND, AST_LOGIC_OR, AST_LOGIC_NOT	The logic operators &&, || and !
AST_TERNARY	The ternary ?:-operator
AST_MEMRD AST_MEMWR	Read and write memories. These nodes are generated by the AST simplifier for writes/reads to/from Verilog arrays.
AST_ASSIGN	An **assign** statement
AST_CELL	A cell instantiation
AST_PRIMITIVE	A primitive cell (**and**, **nand**, **or**, etc.)
AST_ALWAYS, AST_INITIAL	Verilog **always**- and **initial**-blocks
AST_BLOCK	A **begin**-**end**-block
AST_ASSIGN_EQ. AST_ASSIGN_LE	Blocking (=) and nonblocking (<=) assignments within an **always**- or **initial**-block
AST_CASE. AST_COND, AST_DEFAULT	The **case** (**if**) statements, conditions within a case and the default case respectively
AST_FOR	A **for**-loop with an **always**- or **initial**-block
AST_GENVAR, AST_GENBLOCK, AST_GENFOR, AST_GENIF	The **genvar** and **generate** keywords and **for** and **if** within a generate block.
AST_POSEDGE, AST_NEGEDGE, AST_EDGE	Event conditions for **always** blocks.

Table 7.1: AST node types with their corresponding Verilog constructs. (continuation from previous page)

- Inline all task and function calls.

- Evaluate all **generate**-statements and unroll all **for**-loops.

- Perform const folding where it is necessary (e.g. in the value part of AST_PARAMETER, AST_LOCALPARAM, AST_PARASET and AST_RANGE nodes).

- Replace AST_PRIMITIVE nodes with appropriate AST_ASSIGN nodes.

- Replace dynamic bit ranges in the left-hand-side of assignments with AST_CASE nodes with AST_COND children for each possible case.

- Detect array access patterns that are too complicated for the `RTLIL::Memory` abstraction and replace them with a set of signals and cases for all reads and/or writes.

- Otherwise replace array accesses with AST_MEMRD and AST_MEMWR nodes.

In addition to these transformations, the simplifier also annotates the AST with additional information that is needed for the RTLIL generator, namely:

- All ranges (width of signals and bit selections) are not only const folded but (when a constant value is found) are also written to member variables in the AST_RANGE node.

- All identifiers are resolved and all `AST_IDENTIFIER` nodes are annotated with a pointer to the AST node that contains the declaration of the identifier. If no declaration has been found, an AST_AUTOWIRE node is created and used for the annotation.

This produces an AST that is fairly easy to convert to the RTLIL format.

7.2.2 Generating RTLIL

After AST simplification, the `AST::AstNode::genRTLIL()` method of each AST_MODULE node in the AST is called. This initiates a recursive process that generates equivalent RTLIL data for the AST data.

The `AST::AstNode::genRTLIL()` method returns an `RTLIL::SigSpec` structure. For nodes that represent expressions (operators, constants, signals, etc.), the cells needed to implement the calculation described by the expression are created and the resulting signal is returned. That way it is easy to generate the circuits for large expressions using depth-first recursion. For nodes that do not represent an expression (such as AST_CELL), the corresponding circuit is generated and an empty `RTLIL::SigSpec` is returned.

7.3 Synthesizing Verilog always Blocks

For behavioural Verilog code (code utilizing **always**- and **initial**-blocks) it is necessary to also generate `RTLIL::Process` objects. This is done in the following way:

- Whenever `AST::AstNode::genRTLIL()` encounters an **always**- or **initial**-block, it creates an instance of AST_INTERNAL::`ProcessGenerator`. This object then generates the `RTLIL::Process` object for the block. It also calls `AST::AstNode::genRTLIL()` for all right-hand-side expressions contained within the block.

- First the AST_INTERNAL::`ProcessGenerator` creates a list of all signals assigned within the block. It then creates a set of temporary signals using the naming scheme $*<number>**<original_name>* for each of the assigned signals.

- Then an `RTLIL::Process` is created that assigns all intermediate values for each left-hand-side signal to the temporary signal in its `RTLIL::CaseRule`/`RTLIL::SwitchRule` tree.

- Finally a `RTLIL::SyncRule` is created for the `RTLIL::Process` that assigns the temporary signals for the final values to the actual signals.

- Calls to `AST::AstNode::genRTLIL()` are generated for right hand sides as needed. When blocking assignments are used, `AST::AstNode::genRTLIL()` is configured using global variables to use the temporary signals that hold the correct intermediate values whenever one of the previously assigned signals is used in an expression.

Unfortunately the generation of a correct RTLIL::CaseRule/RTLIL::SwitchRule tree for behavioural code is a non-trivial task. The AST frontend solves the problem using the approach described on the following pages. The following example illustrates what the algorithm is supposed to do. Consider the following Verilog code:

```verilog
 1  always @(posedge clock) begin
 2          out1 = in1;
 3          if (in2)
 4                  out1 = !out1;
 5          out2 <= out1;
 6          if (in3)
 7                  out2 <= out2;
 8          if (in4)
 9                  if (in5)
10                          out3 <= in6;
11                  else
12                          out3 <= in7;
13          out1 = out1 ^ out2;
14  end
```

This is translated by the Verilog and AST frontends into the following RTLIL code (attributes, cell parameters and wire declarations not included):

```
 1  cell $logic_not $logic_not$<input>:4$2
 2    connect \A \in1
 3    connect \Y $logic_not$<input>:4$2_Y
 4  end
 5  cell $xor $xor$<input>:13$3
 6    connect \A $1\out1[0:0]
 7    connect \B \out2
 8    connect \Y $xor$<input>:13$3_Y
 9  end
10  process $proc$<input>:1$1
11    assign $0\out3[0:0] \out3
12    assign $0\out2[0:0] $1\out1[0:0]
13    assign $0\out1[0:0] $xor$<input>:13$3_Y
14    switch \in2
15      case 1'1
16        assign $1\out1[0:0] $logic_not$<input>:4$2_Y
17      case
18        assign $1\out1[0:0] \in1
19    end
20    switch \in3
21      case 1'1
22        assign $0\out2[0:0] \out2
23      case
24    end
25    switch \in4
26      case 1'1
27        switch \in5
28          case 1'1
29            assign $0\out3[0:0] \in6
30          case
31            assign $0\out3[0:0] \in7
32        end
```

```
33        case
34      end
35    sync posedge \clock
36        update \out1 $0\out1[0:0]
37        update \out2 $0\out2[0:0]
38        update \out3 $0\out3[0:0]
39  end
```

Note that the two operators are translated into separate cells outside the generated process. The signal out1 is assigned using blocking assignments and therefore out1 has been replaced with a different signal in all expressions after the initial assignment. The signal out2 is assigned using nonblocking assignments and therefore is not substituted on the right-hand-side expressions.

The RTLIL::CaseRule/RTLIL::SwitchRule tree must be interpreted the following way:

- On each case level (the body of the process is the *root case*), first the actions on this level are evaluated and then the switches within the case are evaluated. (Note that the last assignment on line 13 of the Verilog code has been moved to the beginning of the RTLIL process to line 13 of the RTLIL listing.)

  I.e. the special cases deeper in the switch hierarchy override the defaults on the upper levels. The assignments in lines 12 and 22 of the RTLIL code serve as an example for this.

  Note that in contrast to this, the order within the RTLIL::SwitchRule objects within a RTLIL::CaseRule is preserved with respect to the original AST and Verilog code.

- The whole RTLIL::CaseRule/RTLIL::SwitchRule tree describes an asynchronous circuit. I.e. the decision tree formed by the switches can be seen independently for each assigned signal. Whenever one assigned signal changes, all signals that depend on the changed signals are to be updated. For example the assignments in lines 16 and 18 in the RTLIL code in fact influence the assignment in line 12, even though they are in the "wrong order".

The only synchronous part of the process is in the RTLIL::SyncRule object generated at line 35 in the RTLIL code. The sync rule is the only part of the process where the original signals are assigned. The synchronization event from the original Verilog code has been translated into the synchronization type (posedge) and signal (\clock) for the RTLIL::SyncRule object. In the case of this simple example the RTLIL::SyncRule object is later simply transformed into a set of d-type flip-flops and the RTLIL::CaseRule/RTLIL::SwitchRule tree to a decision tree using multiplexers.

In more complex examples (e.g. asynchronous resets) the part of the RTLIL::CaseRule/RTLIL::SwitchRule tree that describes the asynchronous reset must first be transformed to the correct RTLIL::SyncRule objects. This is done by the proc_adff pass.

7.3.1 The ProcessGenerator Algorithm

The AST_INTERNAL::ProcessGenerator uses the following internal state variables:

- subst_rvalue_from and subst_rvalue_to
  These two variables hold the replacement pattern that should be used by AST::AstNode::genRTLIL() for signals with blocking assignments. After initialization of AST_INTERNAL::ProcessGenerator these two variables are empty.

- subst_lvalue_from and subst_lvalue_to
  These two variables contain the mapping from left-hand-side signals (\<name>) to the current temporary signal for the same thing (initially $0\<name>).

- `current_case`
  A pointer to a `RTLIL::CaseRule` object. Initially this is the root case of the generated `RTLIL::Process`.

As the algorithm runs these variables are continuously modified as well as pushed to the stack and later restored to their earlier values by popping from the stack.

On startup the ProcessGenerator generates a new `RTLIL::Process` object with an empty root case and initializes its state variables as described above. Then the `RTLIL::SyncRule` objects are created using the synchronization events from the AST_ALWAYS node and the initial values of `subst_lvalue_from` and `subst_lvalue_to`. Then the AST for this process is evaluated recursively.

During this recursive evaluation, three different relevant types of AST nodes can be discovered: AST_ASSIGN_LE (nonblocking assignments), AST_ASSIGN_EQ (blocking assignments) and AST_CASE (**if** or **case** statement).

7.3.1.1 Handling of Nonblocking Assignments

When an AST_ASSIGN_LE node is discovered, the following actions are performed by the ProcessGenerator:

- The left-hand-side is evaluated using `AST::AstNode::genRTLIL()` and mapped to a temporary signal name using `subst_lvalue_from` and `subst_lvalue_to`.

- The right-hand-side is evaluated using `AST::AstNode::genRTLIL()`. For this call, the values of `subst_rvalue_from` and `subst_rvalue_to` are used to map blocking-assigned signals correctly.

- Remove all assignments to the same left-hand-side as this assignment from the `current_case` and all cases within it.

- Add the new assignment to the `current_case`.

7.3.1.2 Handling of Blocking Assignments

When an AST_ASSIGN_EQ node is discovered, the following actions are performed by the ProcessGenerator:

- Perform all the steps that would be performed for a nonblocking assignment (see above).

- Remove the found left-hand-side (before lvalue mapping) from `subst_rvalue_from` and also remove the respective bits from `subst_rvalue_to`.

- Append the found left-hand-side (before lvalue mapping) to `subst_rvalue_from` and append the found right-hand-side to `subst_rvalue_to`.

7.3.1.3 Handling of Cases and if-Statements

When an AST_CASE node is discovered, the following actions are performed by the ProcessGenerator:

- The values of `subst_rvalue_from`, `subst_rvalue_to`, `subst_lvalue_from` and `subst_lvalue_to` are pushed to the stack.

- A new `RTLIL::SwitchRule` object is generated, the selection expression is evaluated using `AST::AstNode::genRTLIL()` (with the use of `subst_rvalue_from` and `subst_rvalue_to`) and added to the `RTLIL::SwitchRule` object and the object is added to the `current_case`.

- All lvalues assigned to within the AST_CASE node using blocking assignments are collected and saved in the local variable `this_case_eq_lvalue`.

- New temporary signals are generated for all signals in `this_case_eq_lvalue` and stored in `this_case_eq_ltemp`.

- The signals in `this_case_eq_lvalue` are mapped using `subst_rvalue_from` and `subst_rvalue_to` and the resulting set of signals is stored in `this_case_eq_rvalue`.

Then the following steps are performed for each AST_COND node within the AST_CASE node:

- Set `subst_rvalue_from`, `subst_rvalue_to`, `subst_lvalue_from` and `subst_lvalue_to` to the values that have been pushed to the stack.

- Remove `this_case_eq_lvalue` from `subst_lvalue_from`/`subst_lvalue_to`.

- Append `this_case_eq_lvalue` to `subst_lvalue_from` and append `this_case_eq_ltemp` to `subst_lvalue_to`.

- Push the value of `current_case`.

- Create a new RTLIL::CaseRule. Set `current_case` to the new object and add the new object to the RTLIL::SwitchRule created above.

- Add an assignment from `this_case_eq_rvalue` to `this_case_eq_ltemp` to the new `current_case`.

- Evaluate the compare value for this case using AST::AstNode::genRTLIL() (with the use of `subst_rvalue_from` and `subst_rvalue_to`) modify the new `current_case` accordingly.

- Recursion into the children of the AST_COND node.

- Restore `current_case` by popping the old value from the stack.

Finally the following steps are performed:

- The values of `subst_rvalue_from`, `subst_rvalue_to`, `subst_lvalue_from` and `subst_lvalue_to` are popped from the stack.

- The signals from `this_case_eq_lvalue` are removed from the `subst_rvalue_from`/`subst_rvalue_to`-pair.

- The value of `this_case_eq_lvalue` is appended to `subst_rvalue_from` and the value of `this_case_eq_ltemp` is appended to `subst_rvalue_to`.

- Map the signals in `this_case_eq_lvalue` using `subst_lvalue_from`/`subst_lvalue_to`.

- Remove all assignments to signals in `this_case_eq_lvalue` in `current_case` and all cases within it.

- Add an assignment from `this_case_eq_ltemp` to `this_case_eq_lvalue` to `current_case`.

7.3.1.4 Further Analysis of the Algorithm for Cases and if-Statements

With respect to nonblocking assignments the algorithm is easy: later assignments invalidate earlier assignments. For each signal assigned using nonblocking assignments exactly one temporary variable is generated (with the $0-prefix) and this variable is used for all assignments of the variable.

Note how all the _eq_-variables become empty when no blocking assignments are used and many of the steps in the algorithm can then be ignored as a result of this.

For a variable with blocking assignments the algorithm shows the following behaviour: First a new temporary variable is created. This new temporary variable is then registered as the assignment target for all assignments for this variable within the cases for this AST_CASE node. Then for each case the new temporary variable is first assigned the old temporary variable. This assignment is overwritten if the variable is actually assigned in this case and is kept as a default value otherwise.

This yields an RTLIL::CaseRule that assigns the new temporary variable in all branches. So when all cases have been processed a final assignment is added to the containing block that assigns the new temporary variable to the old one. Note how this step always overrides a previous assignment to the old temporary variable. Other than nonblocking assignments, the old assignment could still have an effect somewhere in the design, as there have been calls to AST::AstNode::genRTLIL() with a subst_rvalue_from/subst_rvalue_to-tuple that contained the right-hand-side of the old assignment.

7.3.2 The proc pass

The ProcessGenerator converts a behavioural model in AST representation to a behavioural model in RTLIL::Process representation. The actual conversion from a behavioural model to an RTL representation is performed by the proc pass and the passes it launches:

- proc_clean and proc_rmdead
  These two passes just clean up the RTLIL::Process structure. The proc_clean pass removes empty parts (eg. empty assignments) from the process and proc_rmdead detects and removes unreachable branches from the process's decision trees.

- proc_arst
  This pass detects processes that describe d-type flip-flops with asynchronous resets and rewrites the process to better reflect what they are modelling: Before this pass, an asynchronous reset has two edge-sensitive sync rules and one top-level RTLIL::SwitchRule for the reset path. After this pass the sync rule for the reset is level-sensitive and the top-level RTLIL::SwitchRule has been removed.

- proc_mux
  This pass converts the RTLIL::CaseRule/RTLIL::SwitchRule-tree to a tree of multiplexers per written signal. After this, the RTLIL::Process structure only contains the RTLIL::SyncRules that describe the output registers.

- proc_dff
  This pass replaces the RTLIL::SyncRules to d-type flip-flops (with asynchronous resets if necessary).

- proc_clean
  A final call to proc_clean removes the now empty RTLIL::Process objects.

Performing these last processing steps in passes instead of in the Verilog frontend has two important benefits:

First it improves the transparency of the process. Everything that happens in a separate pass is easier to debug, as the RTLIL data structures can be easily investigated before and after each of the steps.

Second it improves flexibility. This scheme can easily be extended to support other types of storage-elements, such as sr-latches or d-latches, without having to extend the actual Verilog frontend.

7.4 Synthesizing Verilog Arrays

FIXME:
Add some information on the generation of $memrd and $memwr cells and how they are processed in the `memory` pass.

7.5 Synthesizing Parametric Designs

FIXME:
Add some information on the `RTLIL::Module::derive()` method and how it is used to synthesize parametric modules via the `hierarchy` pass.

Chapter 8

Optimizations

Yosys employs a number of optimizations to generate better and cleaner results. This chapter outlines these optimizations.

8.1 Simple Optimizations

The Yosys pass `opt` runs a number of simple optimizations. This includes removing unused signals and cells and const folding. It is recommended to run this pass after each major step in the synthesis script. At the time of this writing the `opt` pass executes the following passes that each perform a simple optimization:

- Once at the beginning of `opt`:
    - `opt_expr`
    - `opt_merge -nomux`
- Repeat until result is stable:
    - `opt_muxtree`
    - `opt_reduce`
    - `opt_merge`
    - `opt_rmdff`
    - `opt_clean`
    - `opt_expr`

The following section describes each of the `opt_*` passes.

8.1.1 The opt_expr pass

This pass performs const folding on the internal combinational cell types described in Chap. 5. This means a cell with all constant inputs is replaced with the constant value this cell drives. In some cases this pass can also optimize cells with some constant inputs.

Table 8.1 shows the replacement rules used for optimizing an $_AND_ gate. The first three rules implement the obvious const folding rules. Note that 'any' might include dynamic values calculated by other parts of the circuit. The following three lines propagate undef (X) states. These are the only three cases in which it is allowed to propagate an undef according to Sec. 5.1.10 of IEEE Std. 1364-2005 [Ver06].

A-Input	B-Input	Replacement
any	0	0
0	any	0
1	1	1
X/Z	X/Z	X
1	X/Z	X
X/Z	1	X
any	X/Z	0
X/Z	any	0
a	1	a
1	b	b

Table 8.1: Const folding rules for $_AND_ cells as used in `opt_expr`.

The next two lines assume the value 0 for undef states. These two rules are only used if no other substitutions are possible in the current module. If other substitutions are possible they are performed first, in the hope that the 'any' will change to an undef value or a 1 and therefore the output can be set to undef.

The last two lines simply replace an $_AND_ gate with one constant-1 input with a buffer.

Besides this basic const folding the `opt_expr` pass can replace 1-bit wide $eq and $ne cells with buffers or not-gates if one input is constant.

The `opt_expr` pass is very conservative regarding optimizing $mux cells, as these cells are often used to model decision-trees and breaking these trees can interfere with other optimizations.

8.1.2 The opt_muxtree pass

This pass optimizes trees of multiplexer cells by analyzing the select inputs. Consider the following simple example:

```
1 module uut(a, y);
2 input a;
3 output [1:0] y = a ? (a ? 1 : 2) : 3;
4 endmodule
```

The output can never be 2, as this would require a to be 1 for the outer multiplexer and 0 for the inner multiplexer. The `opt_muxtree` pass detects this contradiction and replaces the inner multiplexer with a constant 1, yielding the logic for y = a ? 1 : 3.

8.1.3 The opt_reduce pass

This is a simple optimization pass that identifies and consolidates identical input bits to $reduce_and and $reduce_or cells. It also sorts the input bits to ease identification of shareable $reduce_and and $reduce_or cells in other passes.

This pass also identifies and consolidates identical inputs to multiplexer cells. In this case the new shared select bit is driven using a $reduce_or cell that combines the original select bits.

Lastly this pass consolidates trees of $reduce_and cells and trees of $reduce_or cells to single large $reduce_and or $reduce_or cells.

These three simple optimizations are performed in a loop until a stable result is produced.

8.1.4 The opt_rmdff pass

This pass identifies single-bit d-type flip-flops ($\_DFF\_^*$, dff, and $adff$ cells) with a constant data input and replaces them with a constant driver.

8.1.5 The opt_clean pass

This pass identifies unused signals and cells and removes them from the design. It also creates an \unused_bits attribute on wires with unused bits. This attribute can be used for debugging or by other optimization passes.

8.1.6 The opt_merge pass

This pass performs trivial resource sharing. This means that this pass identifies cells with identical inputs and replaces them with a single instance of the cell.

The option -nomux can be used to disable resource sharing for multiplexer cells ($mux and $pmux. This can be useful as it prevents multiplexer trees to be merged, which might prevent opt_muxtree to identify possible optimizations.

8.2 FSM Extraction and Encoding

The fsm pass performs finite-state-machine (FSM) extraction and recoding. The fsm pass simply executes the following other passes:

- Identify and extract FSMs:
  - fsm_detect
  - fsm_extract

- Basic optimizations:
  - fsm_opt
  - opt_clean
  - fsm_opt

- Expanding to nearby gate-logic (if called with -expand):
  - fsm_expand
  - opt_clean
  - fsm_opt

- Re-code FSM states (unless called with -norecode):
  - fsm_recode

- Print information about FSMs:
  - fsm_info

- Export FSMs in KISS2 file format (if called with -export):
  - fsm_export

- Map FSMs to RTL cells (unless called with -nomap):

– `fsm_map`

The `fsm_detect` pass identifies FSM state registers and marks them using the `\fsm_encoding= "auto"` attribute. The `fsm_extract` extracts all FSMs marked using the `\fsm_encoding` attribute (unless `\fsm_encoding` is set to `"none"`) and replaces the corresponding RTL cells with a `$fsm` cell. All other `fsm_*` passes operate on these `$fsm` cells. The `fsm_map` call finally replaces the `$fsm` cells with RTL cells.

Note that these optimizations operate on an RTL netlist. I.e. the `fsm` pass should be executed after the `proc` pass has transformed all `RTLIL::Process` objects to RTL cells.

The algorithms used for FSM detection and extraction are influenced by a more general reported technique [STGR10].

8.2.1 FSM Detection

The `fsm_detect` pass identifies FSM state registers. It sets the `\fsm_encoding= "auto"` attribute on any (multi-bit) wire that matches the following description:

- Does not already have the `\fsm_encoding` attribute.

- Is not an output of the containing module.

- Is driven by single `$dff` or `$adff` cell.

- The `\D`-Input of this `$dff` or `$adff` cell is driven by a multiplexer tree that only has constants or the old state value on its leaves.

- The state value is only used in the said multiplexer tree or by simple relational cells that compare the state value to a constant (usually `$eq` cells).

This heuristic has proven to work very well. It is possible to overwrite it by setting `\fsm_encoding= "auto"` on registers that should be considered FSM state registers and setting `\fsm_encoding= "none"` on registers that match the above criteria but should not be considered FSM state registers.

Note however that marking state registers with `\fsm_encoding` that are not suitable for FSM recoding can cause synthesis to fail or produce invalid results.

8.2.2 FSM Extraction

The `fsm_extract` pass operates on all state signals marked with the `\fsm_encoding` (`!= "none"`) attribute. For each state signal the following information is determined:

- The state registers

- The asynchronous reset state if the state registers use asynchronous reset

- All states and the control input signals used in the state transition functions

- The control output signals calculated from the state signals and control inputs

- A table of all state transitions and corresponding control inputs- and outputs

The state registers (and asynchronous reset state, if applicable) is simply determined by identifying the driver for the state signal.

From there the $mux-tree driving the state register inputs is recursively traversed. All select inputs are control signals and the leaves of the $mux-tree are the states. The algorithm fails if a non-constant leaf that is not the state signal itself is found.

The list of control outputs is initialized with the bits from the state signal. It is then extended by adding all values that are calculated by cells that compare the state signal with a constant value.

In most cases this will cover all uses of the state register, thus rendering the state encoding arbitrary. If however a design uses e.g. a single bit of the state value to drive a control output directly, this bit of the state signal will be transformed to a control output of the same value.

Finally, a transition table for the FSM is generated. This is done by using the `ConstEval` C++ helper class (defined in `kernel/consteval.h`) that can be used to evaluate parts of the design. The `ConstEval` class can be asked to calculate a given set of result signals using a set of signal-value assignments. It can also be passed a list of stop-signals that abort the `ConstEval` algorithm if the value of a stop-signal is needed in order to calculate the result signals.

The `fsm_extract` pass uses the `ConstEval` class in the following way to create a transition table. For each state:

1. Create a `ConstEval` object for the module containing the FSM

2. Add all control inputs to the list of stop signals

3. Set the state signal to the current state

4. Try to evaluate the next state and control output

5. If step 4 was not successful:
   - Recursively goto step 4 with the offending stop-signal set to 0.
   - Recursively goto step 4 with the offending stop-signal set to 1.

6. If step 4 was successful: Emit transition

Finally a $fsm cell is created with the generated transition table and added to the module. This new cell is connected to the control signals and the old drivers for the control outputs are disconnected.

8.2.3 FSM Optimization

The `fsm_opt` pass performs basic optimizations on $fsm cells (not including state recoding). The following optimizations are performed (in this order):

- Unused control outputs are removed from the $fsm cell. The attribute `\unused_bits` (that is usually set by the `opt_clean` pass) is used to determine which control outputs are unused.

- Control inputs that are connected to the same driver are merged.

- When a control input is driven by a control output, the control input is removed and the transition table altered to give the same performance without the external feedback path.

- Entries in the transition table that yield the same output and only differ in the value of a single control input bit are merged and the different bit is removed from the sensitivity list (turned into a don't-care bit).

- Constant inputs are removed and the transition table is altered to give an unchanged behaviour.

- Unused inputs are removed.

8.2.4 FSM Recoding

The `fsm_recode` pass assigns new bit pattern to the states. Usually this also implies a change in the width of the state signal. At the moment of this writing only one-hot encoding with all-zero for the reset state is supported.

The `fsm_recode` pass can also write a text file with the changes performed by it that can be used when verifying designs synthesized by Yosys using Synopsys Formality [24].

8.3 Logic Optimization

Yosys can perform multi-level combinational logic optimization on gate-level netlists using the external program ABC [27]. The `abc` pass extracts the combinational gate-level parts of the design, passes it through ABC, and re-integrates the results. The `abc` pass can also be used to perform other operations using ABC, such as technology mapping (see Sec. 9.3 for details).

Chapter 9

Technology Mapping

Previous chapters outlined how HDL code is transformed into an RTL netlist. The RTL netlist is still based on abstract coarse-grain cell types like arbitrary width adders and even multipliers. This chapter covers how an RTL netlist is transformed into a functionally equivalent netlist utilizing the cell types available in the target architecture.

Technology mapping is often performed in two phases. In the first phase RTL cells are mapped to an internal library of single-bit cells (see Sec. 5.2). In the second phase this netlist of internal gate types is transformed to a netlist of gates from the target technology library.

When the target architecture provides coarse-grain cells (such as block ram or ALUs), these must be mapped to directly form the RTL netlist, as information on the coarse-grain structure of the design is lost when it is mapped to bit-width gate types.

9.1   Cell Substitution

The simplest form of technology mapping is cell substitution, as performed by the `techmap` pass. This pass, when provided with a Verilog file that implements the RTL cell types using simpler cells, simply replaces the RTL cells with the provided implementation.

When no map file is provided, `techmap` uses a built-in map file that maps the Yosys RTL cell types to the internal gate library used by Yosys. The curious reader may find this map file as `techlibs/common/techmap.v` in the Yosys source tree.

Additional features have been added to `techmap` to allow for conditional mapping of cells (see `help techmap` or Sec. C.126). This can for example be useful if the target architecture supports hardware multipliers for certain bit-widths but not for others.

A usual synthesis flow would first use the `techmap` pass to directly map some RTL cells to coarse-grain cells provided by the target architecture (if any) and then use techmap with the built-in default file to map the remaining RTL cells to gate logic.

9.2   Subcircuit Substitution

Sometimes the target architecture provides cells that are more powerful than the RTL cells used by Yosys. For example a cell in the target architecture that can calculate the absolute-difference of two numbers does not match any single RTL cell type but only combinations of cells.

For these cases Yosys provides the `extract` pass that can match a given set of modules against a design and identify the portions of the design that are identical (i.e. isomorphic subcircuits) to any of the given modules. These matched subcircuits are then replaced by instances of the given modules.

The `extract` pass also finds basic variations of the given modules, such as swapped inputs on commutative cell types.

In addition to this the `extract` pass also has limited support for frequent subcircuit mining, i.e. the process of finding recurring subcircuits in the design. This has a few applications, including the design of new coarse-grain architectures [GW13].

The hard algorithmic work done by the `extract` pass (solving the isomorphic subcircuit problem and frequent subcircuit mining) is performed using the SubCircuit library that can also be used stand-alone without Yosys (see Sec. A.3).

9.3 Gate-Level Technology Mapping

On the gate-level the target architecture is usually described by a "Liberty file". The Liberty file format is an industry standard format that can be used to describe the behaviour and other properties of standard library cells [25].

Mapping a design utilizing the Yosys internal gate library (e.g. as a result of mapping it to this representation using the `techmap` pass) is performed in two phases.

First the register cells must be mapped to the registers that are available on the target architectures. The target architecture might not provide all variations of d-type flip-flops with positive and negative clock edge, high-active and low-active asynchronous set and/or reset, etc. Therefore the process of mapping the registers might add additional inverters to the design and thus it is important to map the register cells first.

Mapping of the register cells may be performed by using the `dfflibmap` pass. This pass expects a Liberty file as argument (using the `-liberty` option) and only uses the register cells from the Liberty file.

Secondly the combinational logic must be mapped to the target architecture. This is done using the external program ABC [27] via the `abc` pass by using the `-liberty` option to the pass. Note that in this case only the combinatorial cells are used from the cell library.

Occasionally Liberty files contain trade secrets (such as sensitive timing information) that cannot be shared freely. This complicates processes such as reporting bugs in the tools involved. When the information in the Liberty file used by Yosys and ABC are not part of the sensitive information, the additional tool `yosys-filterlib` (see Sec. B.2) can be used to strip the sensitive information from the Liberty file.

Appendix A

Auxiliary Libraries

The Yosys source distribution contains some auxiliary libraries that are bundled with Yosys.

A.1   SHA1

The files in `libs/sha1/` provide a public domain SHA1 implementation written by Steve Reid, Bruce Guenter, and Volker Grabsch. It is used for generating unique names when specializing parameterized modules.

A.2   BigInt

The files in `libs/bigint/` provide a library for performing arithmetic with arbitrary length integers. It is written by Matt McCutchen [29].

The BigInt library is used for evaluating constant expressions, e.g. using the `ConstEval` class provided in `kernel/consteval.h`.

A.3   SubCircuit

The files in `libs/subcircuit` provide a library for solving the subcircuit isomorphism problem. It is written by Clifford Wolf and based on the Ullmann Subgraph Isomorphism Algorithm [Ull76]. It is used by the `extract` pass (see `help extract` or Sec. C.39).

A.4   ezSAT

The files in `libs/ezsat` provide a library for simplifying generating CNF formulas for SAT solvers. It also contains bindings of MiniSAT. The ezSAT library is written by Clifford Wolf. It is used by the `sat` pass (see `help sat` or Sec. C.102).

Appendix B

Auxiliary Programs

Besides the main `yosys` executable, the Yosys distribution contains a set of additional helper programs.

B.1   yosys-config

The `yosys-config` tool (an auto-generated shell-script) can be used to query compiler options and other information needed for building loadable modules for Yosys. FIXME: See Sec. 6 for details.

B.2   yosys-filterlib

The `yosys-filterlib` tool is a small utility that can be used to strip or extract information from a Liberty file. See Sec. 9.3 for details.

B.3   yosys-abc

This is a unmodified copy of ABC [27]. Not all versions of Yosys work with all versions of ABC. So Yosys comes with its own yosys-abc to avoid compatibility issues between the two.

Appendix C

Command Reference Manual

C.1   abc – use ABC for technology mapping

```
 1      abc [options] [selection]
 2
 3  This pass uses the ABC tool [1] for technology mapping of yosys's internal gate
 4  library to a target architecture.
 5
 6      -exe <command>
 7          use the specified command instead of "<yosys-bindir>/yosys-abc" to execute ABC
 8          This can e.g. be used to call a specific version of ABC or a wrapper.
 9
10      -script <file>
11          use the specified ABC script file instead of the default script.
12
13          if <file> starts with a plus sign (+), then the rest of the filename
14          string is interpreted as the command string to be passed to ABC. The
15          leading plus sign is removed and all commas (,) in the string are
16          replaced with blanks before the string is passed to ABC.
17
18          if no -script parameter is given, the following scripts are used:
19
20          for -liberty without -constr:
21            strash; dc2; scorr; ifraig; retime -o {D}; strash; dch -f;
22                  map {D}
23
24          for -liberty with -constr:
25            strash; dc2; scorr; ifraig; retime -o {D}; strash; dch -f;
26                  map {D}; buffer; upsize {D}; dnsize {D}; stime -p
27
28          for -lut/-luts (only one LUT size):
29            strash; dc2; scorr; ifraig; retime -o; strash; dch -f; if; mfs;
30                  lutpack
31
32          for -lut/-luts (different LUT sizes):
33            strash; dc2; scorr; ifraig; retime -o; strash; dch -f; if; mfs
34
35          for -sop:
```

```
36            strash; dc2; scorr; ifraig; retime -o; strash; dch -f;
37                cover {I} {P}
38
39        otherwise:
40            strash; dc2; scorr; ifraig; retime -o; strash; dch -f; map
41
42    -fast
43        use different default scripts that are slightly faster (at the cost
44        of output quality):
45
46        for -liberty without -constr:
47            retime -o {D}; map {D}
48
49        for -liberty with -constr:
50            retime -o {D}; map {D}; buffer; upsize {D}; dnsize {D}; stime -p
51
52        for -lut/-luts:
53            retime -o; if
54
55        for -sop:
56            retime -o; cover -I {I} -P {P}
57
58        otherwise:
59            retime -o; map
60
61    -liberty <file>
62        generate netlists for the specified cell library (using the liberty
63        file format).
64
65    -constr <file>
66        pass this file with timing constraints to ABC. use with -liberty.
67
68        a constr file contains two lines:
69            set_driving_cell <cell_name>
70            set_load <floating_point_number>
71
72        the set_driving_cell statement defines which cell type is assumed to
73        drive the primary inputs and the set_load statement sets the load in
74        femtofarads for each primary output.
75
76    -D <picoseconds>
77        set delay target. the string {D} in the default scripts above is
78        replaced by this option when used, and an empty string otherwise.
79
80    -I <num>
81        maximum number of SOP inputs.
82        (replaces {I} in the default scripts above)
83
84    -P <num>
85        maximum number of SOP products.
86        (replaces {P} in the default scripts above)
87
88    -lut <width>
89        generate netlist using luts of (max) the specified width.
```

```
 90
 91        -lut <w1>:<w2>
 92            generate netlist using luts of (max) the specified width <w2>. All
 93            luts with width <= <w1> have constant cost. for luts larger than <w1>
 94            the area cost doubles with each additional input bit. the delay cost
 95            is still constant for all lut widths.
 96
 97        -luts <cost1>,<cost2>,<cost3>,<sizeN>:<cost4-N>,..
 98            generate netlist using luts. Use the specified costs for luts with 1,
 99            2, 3, .. inputs.
100
101        -sop
102            map to sum-of-product cells and inverters
103
104        -g type1,type2,...
105            Map the the specified list of gate types. Supported gates types are:
106            AND, NAND, OR, NOR, XOR, XNOR, MUX, AOI3, OAI3, AOI4, OAI4.
107            (The NOT gate is always added to this list automatically.)
108
109        -dff
110            also pass $_DFF_?_ and $_DFFE_??_ cells through ABC. modules with many
111            clock domains are automatically partitioned in clock domains and each
112            domain is passed through ABC independently.
113
114        -clk [!]<clock-signal-name>[,[!]<enable-signal-name>]
115            use only the specified clock domain. this is like -dff, but only FF
116            cells that belong to the specified clock domain are used.
117
118        -keepff
119            set the "keep" attribute on flip-flop output wires. (and thus preserve
120            them, for example for equivalence checking.)
121
122        -nocleanup
123            when this option is used, the temporary files created by this pass
124            are not removed. this is useful for debugging.
125
126        -showtmp
127            print the temp dir name in log. usually this is suppressed so that the
128            command output is identical across runs.
129
130        -markgroups
131            set a 'abcgroup' attribute on all objects created by ABC. The value of
132            this attribute is a unique integer for each ABC process started. This
133            is useful for debugging the partitioning of clock domains.
134
135 When neither -liberty nor -lut is used, the Yosys standard cell library is
136 loaded into ABC before the ABC script is executed.
137
138 This pass does not operate on modules with unprocessed processes in it.
139 (I.e. the 'proc' pass should be used first to convert processes to netlists.)
140
141 [1] http://www.eecs.berkeley.edu/~alanmi/abc/
```

C.2   add – add objects to the design

```
 1      add <command> [selection]
 2
 3  This command adds objects to the design. It operates on all fully selected
 4  modules. So e.g. 'add -wire foo' will add a wire foo to all selected modules.
 5
 6
 7      add {-wire|-input|-inout|-output} <name> <width> [selection]
 8
 9  Add a wire (input, inout, output port) with the given name and width. The
10  command will fail if the object exists already and has different properties
11  than the object to be created.
12
13
14      add -global_input <name> <width> [selection]
15
16  Like 'add -input', but also connect the signal between instances of the
17  selected modules.
```

C.3   aigmap – map logic to and-inverter-graph circuit

```
 1      aigmap [options] [selection]
 2
 3  Replace all logic cells with circuits made of only $_AND_ and
 4  $_NOT_ cells.
 5
 6     -nand
 7         Enable creation of $_NAND_ cells
```

C.4   alumacc – extract ALU and MACC cells

```
 1      alumacc [selection]
 2
 3  This pass translates arithmetic operations like $add, $mul, $lt, etc. to $alu
 4  and $macc cells.
```

C.5   assertpmux – convert internal signals to module ports

```
 1      assertpmux [options] [selection]
 2
 3  This command adds asserts to the design that assert that all parallel muxes
 4  ($pmux cells) have a maximum of one of their inputs enable at any time.
 5
 6     -noinit
 7         do not enforce the pmux condition during the init state
 8
```

```
 9      -always
10          usually the $pmux condition is only checked when the $pmux output
11          is used be the mux tree it drives. this option will deactivate this
12          additional constrained and check the $pmux condition always.
```

C.6   attrmap – renaming attributes

```
 1      attrmap [options] [selection]
 2
 3  This command renames attributes and/or mapps key/value pairs to
 4  other key/value pairs.
 5
 6      -tocase <name>
 7          Match attribute names case-insensitively and set it to the specified
 8          name.
 9
10      -rename <old_name> <new_name>
11          Rename attributes as specified
12
13      -map <old_name>=<old_value> <new_name>=<new_value>
14          Map key/value pairs as indicated.
15
16      -imap <old_name>=<old_value> <new_name>=<new_value>
17          Like -map, but use case-insensitive match for <old_value> when
18          it is a string value.
19
20      -remove <name>=<value>
21          Remove attributes matching this pattern.
22
23      -modattr
24          Operate on module attributes instead of attributes on wires and cells.
25
26  For example, mapping Xilinx-style "keep" attributes to Yosys-style:
27
28      attrmap -tocase keep -imap keep="true" keep=1 \
29              -imap keep="false" keep=0 -remove keep=0
```

C.7   attrmvcp – move or copy attributes from wires to driving cells

```
 1      attrmvcp [options] [selection]
 2
 3  Move or copy attributes on wires to the cells driving them.
 4
 5      -copy
 6          By default, attributes are moved. This will only add
 7          the attribute to the cell, without removing it from
 8          the wire.
 9
10      -purge
11          If no selected cell consumes the attribute, then it is
```

```
12          left on the wire by default. This option will cause the
13          attribute to be removed from the wire, even if no selected
14          cell takes it.
15
16      -driven
17          By default, attriburtes are moved to the cell driving the
18          wire. With this option set it will be moved to the cell
19          driven by the wire instead.
20
21      -attr <attrname>
22          Move or copy this attribute. This option can be used
23          multiple times.
```

C.8   cd – a shortcut for 'select -module <name>'

```
1       cd <modname>
2
3  This is just a shortcut for 'select -module <modname>'.
4
5
6       cd <cellname>
7
8  When no module with the specified name is found, but there is a cell
9  with the specified name in the current module, then this is equivalent
10 to 'cd <celltype>'.
11
12      cd ..
13
14 This is just a shortcut for 'select -clear'.
```

C.9   check – check for obvious problems in the design

```
1       check [options] [selection]
2
3  This pass identifies the following problems in the current design:
4
5   - combinatorial loops
6
7   - two or more conflicting drivers for one wire
8
9   - used wires that do not have a driver
10
11 When called with -noinit then this command also checks for wires which have
12 the 'init' attribute set.
13
14 When called with -assert then the command will produce an error if any
15 problems are found in the current design.
```

C.10   chparam – re-evaluate modules with new parameters

```
1       chparam [ -set name value ]... [selection]
2
3  Re-evaluate the selected modules with new parameters. String values must be
4  passed in double quotes (").
5
6
7       chparam -list [selection]
8
9  List the available parameters of the selected modules.
```

C.11   clean – remove unused cells and wires

```
1       clean [options] [selection]
2
3  This is identical to 'opt_clean', but less verbose.
4
5  When commands are separated using the ';;' token, this command will be executed
6  between the commands.
7
8  When commands are separated using the ';;;' token, this command will be executed
9  in -purge mode between the commands.
```

C.12   clk2fflogic – convert clocked FFs to generic $ff cells

```
1       clk2fflogic [options] [selection]
2
3  This command replaces clocked flip-flops with generic $ff cells that use the
4  implicit global clock. This is useful for formal verification of designs with
5  multiple clocks.
```

C.13   connect – create or remove connections

```
1       connect [-nomap] [-nounset] -set <lhs-expr> <rhs-expr>
2
3  Create a connection. This is equivalent to adding the statement 'assign
4  <lhs-expr> = <rhs-expr>;' to the Verilog input. Per default, all existing
5  drivers for <lhs-expr> are unconnected. This can be overwritten by using
6  the -nounset option.
7
8
9       connect [-nomap] -unset <expr>
10
11 Unconnect all existing drivers for the specified expression.
12
13
```

```
14       connect [-nomap] -port <cell> <port> <expr>
15
16  Connect the specified cell port to the specified cell port.
17
18
19  Per default signal alias names are resolved and all signal names are mapped
20  the the signal name of the primary driver. Using the -nomap option deactivates
21  this behavior.
22
23  The connect command operates in one module only. Either only one module must
24  be selected or an active module must be set using the 'cd' command.
25
26  This command does not operate on module with processes.
```

C.14   connwrappers – replace undef values with defined constants

```
1        connwrappers [options] [selection]
2
3   Wrappers are used in coarse-grain synthesis to wrap cells with smaller ports
4   in wrapper cells with a (larger) constant port size. I.e. the upper bits
5   of the wrapper output are signed/unsigned bit extended. This command uses this
6   knowledge to rewire the inputs of the driven cells to match the output of
7   the driving cell.
8
9       -signed <cell_type> <port_name> <width_param>
10      -unsigned <cell_type> <port_name> <width_param>
11          consider the specified signed/unsigned wrapper output
12
13      -port <cell_type> <port_name> <width_param> <sign_param>
14          use the specified parameter to decide if signed or unsigned
15
16  The options -signed, -unsigned, and -port can be specified multiple times.
```

C.15   copy – copy modules in the design

```
1        copy old_name new_name
2
3   Copy the specified module. Note that selection patterns are not supported
4   by this command.
```

C.16   cover – print code coverage counters

```
1        cover [options] [pattern]
2
3   Print the code coverage counters collected using the cover() macro in the Yosys
4   C++ code. This is useful to figure out what parts of Yosys are utilized by a
5   test bench.
```

```
 6
 7      -q
 8          Do not print output to the normal destination (console and/or log file)
 9
10      -o file
11          Write output to this file, truncate if exists.
12
13      -a file
14          Write output to this file, append if exists.
15
16      -d dir
17          Write output to a newly created file in the specified directory.
18
19  When one or more pattern (shell wildcards) are specified, then only counters
20  matching at least one pattern are printed.
21
22
23  It is also possible to instruct Yosys to print the coverage counters on program
24  exit to a file using environment variables:
25
26      YOSYS_COVER_DIR="{dir-name}" yosys {args}
27
28          This will create a file (with an auto-generated name) in this
29          directory and write the coverage counters to it.
30
31      YOSYS_COVER_FILE="{file-name}" yosys {args}
32
33          This will append the coverage counters to the specified file.
34
35
36  Hint: Use the following AWK command to consolidate Yosys coverage files:
37
38      gawk '{ p[$3] = $1; c[$3] += $2; } END { for (i in p)
39        printf "%-60s %10d %s\n", p[i], c[i], i; }' {files} | sort -k3
40
41
42  Coverage counters are only available in Yosys for Linux.
```

C.17   delete – delete objects in the design

```
 1      delete [selection]
 2
 3  Deletes the selected objects. This will also remove entire modules, if the
 4  whole module is selected.
 5
 6
 7      delete {-input|-output|-port} [selection]
 8
 9  Does not delete any object but removes the input and/or output flag on the
10  selected wires, thus 'deleting' module ports.
```

C.18   deminout – demote inout ports to input or output

```
1       deminout [options] [selection]
2
3  "Demote" inout ports to input or output ports, if possible.
```

C.19   design – save, restore and reset current design

```
1       design -reset
2
3  Clear the current design.
4
5
6       design -save <name>
7
8  Save the current design under the given name.
9
10
11      design -stash <name>
12
13 Save the current design under the given name and then clear the current design.
14
15
16      design -push
17
18 Push the current design to the stack and then clear the current design.
19
20
21      design -pop
22
23 Reset the current design and pop the last design from the stack.
24
25
26      design -load <name>
27
28 Reset the current design and load the design previously saved under the given
29 name.
30
31
32      design -copy-from <name> [-as <new_mod_name>] <selection>
33
34 Copy modules from the specified design into the current one. The selection is
35 evaluated in the other design.
36
37
38      design -copy-to <name> [-as <new_mod_name>] [selection]
39
40 Copy modules from the current design into the specified one.
```

C.20   dff2dffe – transform $dff cells to $dffe cells

```
1      dff2dffe [options] [selection]
2
3  This pass transforms $dff cells driven by a tree of multiplexers with one or
4  more feedback paths to $dffe cells. It also works on gate-level cells such as
5  $_DFF_P_, $_DFF_N_ and $_MUX_.
6
7      -unmap
8          operate in the opposite direction: replace $dffe cells with combinations
9          of $dff and $mux cells. the options below are ignore in unmap mode.
10
11     -direct <internal_gate_type> <external_gate_type>
12         map directly to external gate type. <internal_gate_type> can
13         be any internal gate-level FF cell (except $_DFFE_??_). the
14         <external_gate_type> is the cell type name for a cell with an
15         identical interface to the <internal_gate_type>, except it
16         also has an high-active enable port 'E'.
17            Usually <external_gate_type> is an intermediate cell type
18         that is then translated to the final type using 'techmap'.
19
20     -direct-match <pattern>
21         like -direct for all DFF cell types matching the expression.
22         this will use $__DFFE_* as <external_gate_type> matching the
23         internal gate type $_DFF_*_, except for $_DFF_[NP]_, which is
24         converted to $_DFFE_[NP]_.
```

C.21   dffinit – set INIT param on FF cells

```
1      dffinit [options] [selection]
2
3  This pass sets an FF cell parameter to the the initial value of the net it
4  drives. (This is primarily used in FPGA flows.)
5
6      -ff <cell_name> <output_port> <init_param>
7          operate on the specified cell type. this option can be used
8          multiple times.
```

C.22   dfflibmap – technology mapping of flip-flops

```
1      dfflibmap [-prepare] -liberty <file> [selection]
2
3  Map internal flip-flop cells to the flip-flop cells in the technology
4  library specified in the given liberty file.
5
6  This pass may add inverters as needed. Therefore it is recommended to
7  first run this pass and then map the logic paths to the target technology.
8
9  When called with -prepare, this command will convert the internal FF cells
```

```
10  to the internal cell types that best match the cells found in the given
11  liberty file.
```

C.23 dffsr2dff – convert DFFSR cells to simpler FF cell types

```
1       dffsr2dff [options] [selection]
2
3   This pass converts DFFSR cells ($dffsr, $_DFFSR_???_) and ADFF cells ($adff,
4   $_DFF_???_) to simpler FF cell types when any of the set/reset inputs is unused.
```

C.24 dump – print parts of the design in ilang format

```
1       dump [options] [selection]
2
3   Write the selected parts of the design to the console or specified file in
4   ilang format.
5
6       -m
7           also dump the module headers, even if only parts of a single
8           module is selected
9
10      -n
11          only dump the module headers if the entire module is selected
12
13      -o <filename>
14          write to the specified file.
15
16      -a <filename>
17          like -outfile but append instead of overwrite
```

C.25 echo – turning echoing back of commands on and off

```
1       echo on
2
3   Print all commands to log before executing them.
4
5
6       echo off
7
8   Do not print all commands to log before executing them. (default)
```

C.26 edgetypes – list all types of edges in selection

```
1      edgetypes [options] [selection]
2
3  This command lists all unique types of 'edges' found in the selection. An 'edge'
4  is a 4-tuple of source and sink cell type and port name.
```

C.27   equiv_add – add a $equiv cell

```
1      equiv_add [-try] gold_sig gate_sig
2
3  This command adds an $equiv cell for the specified signals.
4
5
6      equiv_add [-try] -cell gold_cell gate_cell
7
8  This command adds $equiv cells for the ports of the specified cells.
```

C.28   equiv_induct – proving $equiv cells using temporal induction

```
1      equiv_induct [options] [selection]
2
3  Uses a version of temporal induction to prove $equiv cells.
4
5  Only selected $equiv cells are proven and only selected cells are used to
6  perform the proof.
7
8      -undef
9          enable modelling of undef states
10
11     -seq <N>
12         the max. number of time steps to be considered (default = 4)
13
14 This command is very effective in proving complex sequential circuits, when
15 the internal state of the circuit quickly propagates to $equiv cells.
16
17 However, this command uses a weak definition of 'equivalence': This command
18 proves that the two circuits will not diverge after they produce equal
19 outputs (observable points via $equiv) for at least <N> cycles (the <N>
20 specified via -seq).
21
22 Combined with simulation this is very powerful because simulation can give
23 you confidence that the circuits start out synced for at least <N> cycles
24 after reset.
```

C.29   equiv_make – prepare a circuit for equivalence checking

```
1       equiv_make [options] gold_module gate_module equiv_module
2
3  This creates a module annotated with $equiv cells from two presumably
4  equivalent modules. Use commands such as 'equiv_simple' and 'equiv_status'
5  to work with the created equivalent checking module.
6
7      -inames
8          Also match cells and wires with $... names.
9
10     -blacklist <file>
11         Do not match cells or signals that match the names in the file.
12
13     -encfile <file>
14         Match FSM encodings using the description from the file.
15         See 'help fsm_recode' for details.
16
17 Note: The circuit created by this command is not a miter (with something like
18 a trigger output), but instead uses $equiv cells to encode the equivalence
19 checking problem. Use 'miter -equiv' if you want to create a miter circuit.
```

C.30   equiv_mark – mark equivalence checking regions

```
1       equiv_mark [options] [selection]
2
3  This command marks the regions in an equivalence checking module. Region 0 is
4  the proven part of the circuit. Regions with higher numbers are connected
5  unproven subcricuits. The integer attribute 'equiv_region' is set on all
6  wires and cells.
```

C.31   equiv_miter – extract miter from equiv circuit

```
1       equiv_miter [options] miter_module [selection]
2
3  This creates a miter module for further analysis of the selected $equiv cells.
4
5      -trigger
6          Create a trigger output
7
8      -cmp
9          Create cmp_* outputs for individual unproven $equiv cells
10
11     -assert
12         Create a $assert cell for each unproven $equiv cell
13
14     -undef
15         Create compare logic that handles undefs correctly
```

86

C.32   equiv_purge – purge equivalence checking module

```
1       equiv_purge [options] [selection]
2
3   This command removes the proven part of an equivalence checking module, leaving
4   only the unproven segments in the design. This will also remove and add module
5   ports as needed.
```

C.33   equiv_remove – remove $equiv cells

```
1       equiv_remove [options] [selection]
2
3   This command removes the selected $equiv cells. If neither -gold nor -gate is
4   used then only proven cells are removed.
5
6       -gold
7           keep gold circuit
8
9       -gate
10          keep gate circuit
```

C.34   equiv_simple – try proving simple $equiv instances

```
1       equiv_simple [options] [selection]
2
3   This command tries to prove $equiv cells using a simple direct SAT approach.
4
5       -v
6           verbose output
7
8       -undef
9           enable modelling of undef states
10
11      -nogroup
12          disabling grouping of $equiv cells by output wire
13
14      -seq <N>
15          the max. number of time steps to be considered (default = 1)
```

C.35   equiv_status – print status of equivalent checking module

```
1       equiv_status [options] [selection]
2
3   This command prints status information for all selected $equiv cells.
4
5       -assert
6           produce an error if any unproven $equiv cell is found
```

C.36 equiv_struct – structural equivalence checking

```
     equiv_struct [options] [selection]

This command adds additional $equiv cells based on the assumption that the
gold and gate circuit are structurally equivalent. Note that this can introduce
bad $equiv cells in cases where the netlists are not structurally equivalent,
for example when analyzing circuits with cells with commutative inputs. This
command will also de-duplicate gates.

    -fwd
        by default this command performans forward sweeps until nothing can
        be merged by forwards sweeps, then backward sweeps until forward
        sweeps are effective again. with this option set only forward sweeps
        are performed.

    -fwonly <cell_type>
        add the specified cell type to the list of cell types that are only
        merged in forward sweeps and never in backward sweeps. $equiv is in
        this list automatically.

    -icells
        by default, the internal RTL and gate cell types are ignored. add
        this option to also process those cell types with this command.

    -maxiter <N>
        maximum number of iterations to run before aborting
```

C.37 eval – evaluate the circuit given an input

```
     eval [options] [selection]

This command evaluates the value of a signal given the value of all required
inputs.

    -set <signal> <value>
        set the specified signal to the specified value.

    -set-undef
        set all unspecified source signals to undef (x)

    -table <signal>
        create a truth table using the specified input signals

    -show <signal>
        show the value for the specified signal. if no -show option is passed
        then all output ports of the current module are used.
```

C.38 expose – convert internal signals to module ports

```
1    expose [options] [selection]
2
3  This command exposes all selected internal signals of a module as additional
4  outputs.
5
6      -dff
7          only consider wires that are directly driven by register cell.
8
9      -cut
10         when exposing a wire, create an input/output pair and cut the internal
11         signal path at that wire.
12
13     -shared
14         only expose those signals that are shared among the selected modules.
15         this is useful for preparing modules for equivalence checking.
16
17     -evert
18         also turn connections to instances of other modules to additional
19         inputs and outputs and remove the module instances.
20
21     -evert-dff
22         turn flip-flops to sets of inputs and outputs.
23
24     -sep <separator>
25         when creating new wire/port names, the original object name is suffixed
26         with this separator (default: '.') and the port name or a type
27         designator for the exposed signal.
```

C.39 extract – find subcircuits and replace them with cells

```
1    extract -map <map_file> [options] [selection]
2    extract -mine <out_file> [options] [selection]
3
4  This pass looks for subcircuits that are isomorphic to any of the modules
5  in the given map file and replaces them with instances of this modules. The
6  map file can be a Verilog source file (*.v) or an ilang file (*.il).
7
8      -map <map_file>
9          use the modules in this file as reference. This option can be used
10         multiple times.
11
12     -map %<design-name>
13         use the modules in this in-memory design as reference. This option can
14         be used multiple times.
15
16     -verbose
17         print debug output while analyzing
18
19     -constports
20         also find instances with constant drivers. this may be much
21         slower than the normal operation.
```

```
22
23      -nodefaultswaps
24          normally builtin port swapping rules for internal cells are used per
25          default. This turns that off, so e.g. 'a^b' does not match 'b^a'
26          when this option is used.
27
28      -compat <needle_type> <haystack_type>
29          Per default, the cells in the map file (needle) must have the
30          type as the cells in the active design (haystack). This option
31          can be used to register additional pairs of types that should
32          match. This option can be used multiple times.
33
34      -swap <needle_type> <port1>,<port2>[,...]
35          Register a set of swappable ports for a needle cell type.
36          This option can be used multiple times.
37
38      -perm <needle_type> <port1>,<port2>[,...] <portA>,<portB>[,...]
39          Register a valid permutation of swappable ports for a needle
40          cell type. This option can be used multiple times.
41
42      -cell_attr <attribute_name>
43          Attributes on cells with the given name must match.
44
45      -wire_attr <attribute_name>
46          Attributes on wires with the given name must match.
47
48      -ignore_parameters
49          Do not use parameters when matching cells.
50
51      -ignore_param <cell_type> <parameter_name>
52          Do not use this parameter when matching cells.
53
54  This pass does not operate on modules with unprocessed processes in it.
55  (I.e. the 'proc' pass should be used first to convert processes to netlists.)
56
57  This pass can also be used for mining for frequent subcircuits. In this mode
58  the following options are to be used instead of the -map option.
59
60      -mine <out_file>
61          mine for frequent subcircuits and write them to the given ilang file
62
63      -mine_cells_span <min> <max>
64          only mine for subcircuits with the specified number of cells
65          default value: 3 5
66
67      -mine_min_freq <num>
68          only mine for subcircuits with at least the specified number of matches
69          default value: 10
70
71      -mine_limit_matches_per_module <num>
72          when calculating the number of matches for a subcircuit, don't count
73          more than the specified number of matches per module
74
75      -mine_max_fanout <num>
```

```
76          don't consider internal signals with more than <num> connections
77
78  The modules in the map file may have the attribute 'extract_order' set to an
79  integer value. Then this value is used to determine the order in which the pass
80  tries to map the modules to the design (ascending, default value is 0).
81
82  See 'help techmap' for a pass that does the opposite thing.
```

C.40   flatten – flatten design

```
1       flatten [selection]
2
3  This pass flattens the design by replacing cells by their implementation. This
4  pass is very similar to the 'techmap' pass. The only difference is that this
5  pass is using the current design as mapping library.
6
7  Cells and/or modules with the 'keep_hierarchy' attribute set will not be
8  flattened by this command.
```

C.41   freduce – perform functional reduction

```
1       freduce [options] [selection]
2
3  This pass performs functional reduction in the circuit. I.e. if two nodes are
4  equivalent, they are merged to one node and one of the redundant drivers is
5  disconnected. A subsequent call to 'clean' will remove the redundant drivers.
6
7      -v, -vv
8          enable verbose or very verbose output
9
10     -inv
11         enable explicit handling of inverted signals
12
13     -stop <n>
14         stop after <n> reduction operations. this is mostly used for
15         debugging the freduce command itself.
16
17     -dump <prefix>
18         dump the design to <prefix>_<module>_<num>.il after each reduction
19         operation. this is mostly used for debugging the freduce command.
20
21 This pass is undef-aware, i.e. it considers don't-care values for detecting
22 equivalent nodes.
23
24 All selected wires are considered for rewiring. The selected cells cover the
25 circuit that is analyzed.
```

C.42 fsm – extract and optimize finite state machines

```
 1      fsm [options] [selection]
 2
 3  This pass calls all the other fsm_* passes in a useful order. This performs
 4  FSM extraction and optimization. It also calls opt_clean as needed:
 5
 6      fsm_detect          unless got option -nodetect
 7      fsm_extract
 8
 9      fsm_opt
10      opt_clean
11      fsm_opt
12
13      fsm_expand          if got option -expand
14      opt_clean           if got option -expand
15      fsm_opt             if got option -expand
16
17      fsm_recode          unless got option -norecode
18
19      fsm_info
20
21      fsm_export          if got option -export
22      fsm_map             unless got option -nomap
23
24  Options:
25
26      -expand, -norecode, -export, -nomap
27          enable or disable passes as indicated above
28
29      -fullexpand
30          call expand with -full option
31
32      -encoding type
33      -fm_set_fsm_file file
34      -encfile file
35          passed through to fsm_recode pass
```

C.43 fsm_detect – finding FSMs in design

```
 1      fsm_detect [selection]
 2
 3  This pass detects finite state machines by identifying the state signal.
 4  The state signal is then marked by setting the attribute 'fsm_encoding'
 5  on the state signal to "auto".
 6
 7  Existing 'fsm_encoding' attributes are not changed by this pass.
 8
 9  Signals can be protected from being detected by this pass by setting the
10  'fsm_encoding' attribute to "none".
```

C.44   fsm_expand – expand FSM cells by merging logic into it

```
1      fsm_expand [-full] [selection]
2
3  The fsm_extract pass is conservative about the cells that belong to a finite
4  state machine. This pass can be used to merge additional auxiliary gates into
5  the finite state machine.
6
7  By default, fsm_expand is still a bit conservative regarding merging larger
8  word-wide cells. Call with -full to consider all cells for merging.
```

C.45   fsm_export – exporting FSMs to KISS2 files

```
1      fsm_export [-noauto] [-o filename] [-origenc] [selection]
2
3  This pass creates a KISS2 file for every selected FSM. For FSMs with the
4  'fsm_export' attribute set, the attribute value is used as filename, otherwise
5  the module and cell name is used as filename. If the parameter '-o' is given,
6  the first exported FSM is written to the specified filename. This overwrites
7  the setting as specified with the 'fsm_export' attribute. All other FSMs are
8  exported to the default name as mentioned above.
9
10     -noauto
11         only export FSMs that have the 'fsm_export' attribute set
12
13     -o filename
14         filename of the first exported FSM
15
16     -origenc
17         use binary state encoding as state names instead of s0, s1, ...
```

C.46   fsm_extract – extracting FSMs in design

```
1      fsm_extract [selection]
2
3  This pass operates on all signals marked as FSM state signals using the
4  'fsm_encoding' attribute. It consumes the logic that creates the state signal
5  and uses the state signal to generate control signal and replaces it with an
6  FSM cell.
7
8  The generated FSM cell still generates the original state signal with its
9  original encoding. The 'fsm_opt' pass can be used in combination with the
10 'opt_clean' pass to eliminate this signal.
```

C.47   fsm_info – print information on finite state machines

```
1       fsm_info [selection]
2
3   This pass dumps all internal information on FSM cells. It can be useful for
4   analyzing the synthesis process and is called automatically by the 'fsm'
5   pass so that this information is included in the synthesis log file.
```

C.48   fsm_map – mapping FSMs to basic logic

```
1       fsm_map [selection]
2
3   This pass translates FSM cells to flip-flops and logic.
```

C.49   fsm_opt – optimize finite state machines

```
1       fsm_opt [selection]
2
3   This pass optimizes FSM cells. It detects which output signals are actually
4   not used and removes them from the FSM. This pass is usually used in
5   combination with the 'opt_clean' pass (see also 'help fsm').
```

C.50   fsm_recode – recoding finite state machines

```
1       fsm_recode [options] [selection]
2
3   This pass reassign the state encodings for FSM cells. At the moment only
4   one-hot encoding and binary encoding is supported.
5       -encoding <type>
6           specify the encoding scheme used for FSMs without the
7           'fsm_encoding' attribute or with the attribute set to 'auto'.
8
9       -fm_set_fsm_file <file>
10          generate a file containing the mapping from old to new FSM encoding
11          in form of Synopsys Formality set_fsm_* commands.
12
13      -encfile <file>
14          write the mappings from old to new FSM encoding to a file in the
15          following format:
16
17              .fsm <module_name> <state_signal>
18              .map <old_bitpattern> <new_bitpattern>
```

C.51   greenpak4_counters – Extract GreenPak4 counter cells

```
1      greenpak4_counters [options] [selection]
2
3  This pass converts non-resettable or async resettable down counters to GreenPak4
4  counter cells (All other GreenPak4 counter modes must be instantiated manually.)
```

C.52   greenpak4_dffinv – merge greenpak4 inverters and DFFs

```
1      greenpak4_dffinv [options] [selection]
2
3  Merge GP_INV cells with GP_DFF* cells.
```

C.53   help – display help messages

```
1      help  ...............  list all commands
2      help <command>  ......  print help message for given command
3      help -all  ..........  print complete command reference
4
5      help -cells .........  list all cell types
6      help <celltype>  .....  print help message for given cell type
7      help <celltype>+  ....  print verilog code for given cell type
```

C.54   hierarchy – check, expand and clean up design hierarchy

```
1      hierarchy [-check] [-top <module>]
2      hierarchy -generate <cell-types> <port-decls>
3
4  In parametric designs, a module might exists in several variations with
5  different parameter values. This pass looks at all modules in the current
6  design an re-runs the language frontends for the parametric modules as
7  needed.
8
9      -check
10         also check the design hierarchy. this generates an error when
11         an unknown module is used as cell type.
12
13     -purge_lib
14         by default the hierarchy command will not remove library (blackbox)
15         modules. use this option to also remove unused blackbox modules.
16
17     -libdir <directory>
18         search for files named <module_name>.v in the specified directory
19         for unknown modules and automatically run read_verilog for each
20         unknown module.
21
22     -keep_positionals
23         per default this pass also converts positional arguments in cells
```

```
24          to arguments using port names. this option disables this behavior.
25
26      -nokeep_asserts
27          per default this pass sets the "keep" attribute on all modules
28          that directly or indirectly contain one or more $assert cells. this
29          option disables this behavior.
30
31      -top <module>
32          use the specified top module to built a design hierarchy. modules
33          outside this tree (unused modules) are removed.
34
35          when the -top option is used, the 'top' attribute will be set on the
36          specified top module. otherwise a module with the 'top' attribute set
37          will implicitly be used as top module, if such a module exists.
38
39      -auto-top
40          automatically determine the top of the design hierarchy and mark it.
41
42  In -generate mode this pass generates blackbox modules for the given cell
43  types (wildcards supported). For this the design is searched for cells that
44  match the given types and then the given port declarations are used to
45  determine the direction of the ports. The syntax for a port declaration is:
46
47      {i|o|io}[@<num>]:<portname>
48
49  Input ports are specified with the 'i' prefix, output ports with the 'o'
50  prefix and inout ports with the 'io' prefix. The optional <num> specifies
51  the position of the port in the parameter list (needed when instantiated
52  using positional arguments). When <num> is not specified, the <portname> can
53  also contain wildcard characters.
54
55  This pass ignores the current selection and always operates on all modules
56  in the current design.
```

C.55 hilomap – technology mapping of constant hi- and/or lo-drivers

```
1       hilomap [options] [selection]
2
3   Map constants to 'tielo' and 'tiehi' driver cells.
4
5       -hicell <celltype> <portname>
6           Replace constant hi bits with this cell.
7
8       -locell <celltype> <portname>
9           Replace constant lo bits with this cell.
10
11      -singleton
12          Create only one hi/lo cell and connect all constant bits
13          to that cell. Per default a separate cell is created for
14          each constant bit.
```

C.56 history – show last interactive commands

```
1       history
2
3   This command prints all commands in the shell history buffer. This are
4   all commands executed in an interactive session, but not the commands
5   from executed scripts.
```

C.57 ice40_ffinit – iCE40: handle FF init values

```
1       ice40_ffinit [options] [selection]
2
3   Remove zero init values for FF output signals. Add inverters to implement
4   nonzero init values.
```

C.58 ice40_ffssr – iCE40: merge synchronous set/reset into FF cells

```
1       ice40_ffssr [options] [selection]
2
3   Merge synchronous set/reset $_MUX_ cells into iCE40 FFs.
```

C.59 ice40_opt – iCE40: perform simple optimizations

```
1       ice40_opt [options] [selection]
2
3   This command executes the following script:
4
5       do
6           <ice40 specific optimizations>
7           opt_expr -mux_undef -undriven [-full]
8           opt_merge
9           opt_rmdff
10          opt_clean
11      while <changed design>
12
13  When called with the option -unlut, this command will transform all already
14  mapped SB_LUT4 cells back to logic.
```

C.60 insbuf – insert buffer cells for connected wires

```
1       insbuf [options] [selection]
2
3   Insert buffer cells into the design for directly connected wires.
4
```

```
5        -buf <celltype> <in-portname> <out-portname>
6            Use the given cell type instead of $_BUF_. (Notice that the next
7            call to "clean" will remove all $_BUF_ in the design.)
```

C.61   iopadmap – technology mapping of i/o pads (or buffers)

```
1        iopadmap [options] [selection]
2
3   Map module inputs/outputs to PAD cells from a library. This pass
4   can only map to very simple PAD cells. Use 'techmap' to further map
5   the resulting cells to more sophisticated PAD cells.
6
7        -inpad <celltype> <portname>[:<portname>]
8            Map module input ports to the given cell type with the
9            given output port name. if a 2nd portname is given, the
10           signal is passed through the pad call, using the 2nd
11           portname as the port facing the module port.
12
13       -outpad <celltype> <portname>[:<portname>]
14       -inoutpad <celltype> <portname>[:<portname>]
15           Similar to -inpad, but for output and inout ports.
16
17       -toutpad <celltype> <portname>:<portname>[:<portname>]
18           Merges $_TBUF_ cells into the output pad cell. This takes precedence
19           over the other -outpad cell. The first portname is the enable input
20           of the tristate driver.
21
22       -tinoutpad <celltype> <portname>:<portname>:<portname>[:<portname>]
23           Merges $_TBUF_ cells into the inout pad cell. This takes precedence
24           over the other -inoutpad cell. The first portname is the enable input
25           of the tristate driver and the 2nd portname is the internal output
26           buffering the external signal.
27
28       -widthparam <param_name>
29           Use the specified parameter name to set the port width.
30
31       -nameparam <param_name>
32           Use the specified parameter to set the port name.
33
34       -bits
35           create individual bit-wide buffers even for ports that
36           are wider. (the default behavior is to create word-wide
37           buffers using -widthparam to set the word size on the cell.)
38
39   Tristate PADS (-toutpad, -tinoutpad) always operate in -bits mode.
```

C.62   json – write design in JSON format

```
1        json [options] [selection]
2
```

```
 3 │Write a JSON netlist of all selected objects.
 4 │
 5 │    -o <filename>
 6 │        write to the specified file.
 7 │
 8 │    -aig
 9 │        also include AIG models for the different gate types
10 │
11 │See 'help write_json' for a description of the JSON format used.
```

C.63   log – print text and log files

```
 1 │    log string
 2 │
 3 │Print the given string to the screen and/or the log file. This is useful for TCL
 4 │scripts, because the TCL command "puts" only goes to stdout but not to
 5 │logfiles.
 6 │
 7 │    -stdout
 8 │        Print the output to stdout too. This is useful when all Yosys is executed
 9 │        with a script and the -q (quiet operation) argument to notify the user.
10 │
11 │    -stderr
12 │        Print the output to stderr too.
13 │
14 │    -nolog
15 │        Don't use the internal log() command. Use either -stdout or -stderr,
16 │        otherwise no output will be generated at all.
17 │
18 │    -n
19 │        do not append a newline
```

C.64   ls – list modules or objects in modules

```
 1 │    ls [selection]
 2 │
 3 │When no active module is selected, this prints a list of modules.
 4 │
 5 │When an active module is selected, this prints a list of objects in the module.
```

C.65   lut2mux – convert $lut to $_MUX_

```
 1 │    lut2mux [options] [selection]
 2 │
 3 │This pass converts $lut cells to $_MUX_ gates.
```

C.66   maccmap – mapping macc cells

```
1       maccmap [-unmap] [selection]
2
3  This pass maps $macc cells to yosys $fa and $alu cells. When the -unmap option
4  is used then the $macc cell is mapped to $add, $sub, etc. cells instead.
```

C.67   memory – translate memories to basic cells

```
1       memory [-nomap] [-nordff] [-memx] [-bram <bram_rules>] [selection]
2
3  This pass calls all the other memory_* passes in a useful order:
4
5      memory_dff [-nordff]               (-memx implies -nordff)
6      opt_clean
7      memory_share
8      opt_clean
9      memory_memx                       (when called with -memx)
10     memory_collect
11     memory_bram -rules <bram_rules>   (when called with -bram)
12     memory_map                        (skipped if called with -nomap)
13
14 This converts memories to word-wide DFFs and address decoders
15 or multiport memory blocks if called with the -nomap option.
```

C.68   memory_bram – map memories to block rams

```
1       memory_bram -rules <rule_file> [selection]
2
3  This pass converts the multi-port $mem memory cells into block ram instances.
4  The given rules file describes the available resources and how they should be
5  used.
6
7  The rules file contains a set of block ram description and a sequence of match
8  rules. A block ram description looks like this:
9
10     bram RAMB1024X32      # name of BRAM cell
11       init 1             # set to '1' if BRAM can be initialized
12       abits 10           # number of address bits
13       dbits 32           # number of data bits
14       groups 2           # number of port groups
15       ports  1 1         # number of ports in each group
16       wrmode 1 0         # set to '1' if this groups is write ports
17       enable 4 1         # number of enable bits
18       transp 0 2         # transparent (for read ports)
19       clocks 1 2         # clock configuration
20       clkpol 2 2         # clock polarity configuration
21     endbram
22
```

For the option 'transp' the value 0 means non-transparent, 1 means transparent
and a value greater than 1 means configurable. All groups with the same
value greater than 1 share the same configuration bit.

For the option 'clocks' the value 0 means non-clocked, and a value greater
than 0 means clocked. All groups with the same value share the same clock
signal.

For the option 'clkpol' the value 0 means negative edge, 1 means positive edge
and a value greater than 1 means configurable. All groups with the same value
greater than 1 share the same configuration bit.

Using the same bram name in different bram blocks will create different variants
of the bram. Verilog configuration parameters for the bram are created as needed.

It is also possible to create variants by repeating statements in the bram block
and appending '@<label>' to the individual statements.

A match rule looks like this:

```
    match RAMB1024X32
      max waste 16384     # only use this bram if <= 16k ram bits are unused
      min efficiency 80   # only use this bram if efficiency is at least 80%
    endmatch
```

It is possible to match against the following values with min/max rules:

```
    words   ........   number of words in memory in design
    abits   ........   number of address bits on memory in design
    dbits   ........   number of data bits on memory in design
    wports  .......    number of write ports on memory in design
    rports  .......    number of read ports on memory in design
    ports   ........   number of ports on memory in design
    bits    .........  number of bits in memory in design
    dups .........     number of duplications for more read ports

    awaste  .......    number of unused address slots for this match
    dwaste  .......    number of unused data bits for this match
    bwaste  .......    number of unused bram bits for this match
    waste   ........   total number of unused bram bits (bwaste*dups)
    efficiency  ...    total percentage of used and non-duplicated bits

    acells  .......    number of cells in 'address-direction'
    dcells  .......    number of cells in 'data-direction'
    cells   ........   total number of cells (acells*dcells*dups)
```

The interface for the created bram instances is derived from the bram
description. Use 'techmap' to convert the created bram instances into
instances of the actual bram cells of your target architecture.

A match containing the command 'or_next_if_better' is only used if it
has a higher efficiency than the next match (and the one after that if
the next also has 'or_next_if_better' set, and so forth).

```
77  A match containing the command 'make_transp' will add external circuitry
78  to simulate 'transparent read', if necessary.
79
80  A match containing the command 'make_outreg' will add external flip-flops
81  to implement synchronous read ports, if necessary.
82
83  A match containing the command 'shuffle_enable A' will re-organize
84  the data bits to accommodate the enable pattern of port A.
```

C.69   memory_collect – creating multi-port memory cells

```
1      memory_collect [selection]
2
3  This pass collects memories and memory ports and creates generic multiport
4  memory cells.
```

C.70   memory_dff – merge input/output DFFs into memories

```
1      memory_dff [options] [selection]
2
3  This pass detects DFFs at memory ports and merges them into the memory port.
4  I.e. it consumes an asynchronous memory port and the flip-flops at its
5  interface and yields a synchronous memory port.
6
7      -nordfff
8          do not merge registers on read ports
```

C.71   memory_map – translate multiport memories to basic cells

```
1      memory_map [selection]
2
3  This pass converts multiport memory cells as generated by the memory_collect
4  pass to word-wide DFFs and address decoders.
```

C.72   memory_memx – emulate vlog sim behavior for mem ports

```
1      memory_memx [selection]
2
3  This pass adds additional circuitry that emulates the Verilog simulation
4  behavior for out-of-bounds memory reads and writes.
```

C.73   memory_share – consolidate memory ports

```
    memory_share [selection]

This pass merges share-able memory ports into single memory ports.

The following methods are used to consolidate the number of memory ports:

  - When write ports are connected to async read ports accessing the same
    address, then this feedback path is converted to a write port with
    byte/part enable signals.

  - When multiple write ports access the same address then this is converted
    to a single write port with a more complex data and/or enable logic path.

  - When multiple write ports are never accessed at the same time (a SAT
    solver is used to determine this), then the ports are merged into a single
    write port.

Note that in addition to the algorithms implemented in this pass, the $memrd
and $memwr cells are also subject to generic resource sharing passes (and other
optimizations) such as "share" and "opt_merge".
```

C.74   memory_unpack – unpack multi-port memory cells

```
    memory_unpack [selection]

This pass converts the multi-port $mem memory cells into individual $memrd and
$memwr cells. It is the counterpart to the memory_collect pass.
```

C.75   miter – automatically create a miter circuit

```
    miter -equiv [options] gold_name gate_name miter_name

Creates a miter circuit for equivalence checking. The gold- and gate- modules
must have the same interfaces. The miter circuit will have all inputs of the
two source modules, prefixed with 'in_'. The miter circuit has a 'trigger'
output that goes high if an output mismatch between the two source modules is
detected.

    -ignore_gold_x
        a undef (x) bit in the gold module output will match any value in
        the gate module output.

    -make_outputs
        also route the gold- and gate-outputs to 'gold_*' and 'gate_*' outputs
        on the miter circuit.

    -make_outcmp
```

```
18          also create a cmp_* output for each gold/gate output pair.
19
20      -make_assert
21          also create an 'assert' cell that checks if trigger is always low.
22
23      -flatten
24          call 'flatten; opt_expr -keepdc -undriven;;' on the miter circuit.
25
26
27      miter -assert [options] module [miter_name]
28
29  Creates a miter circuit for property checking. All input ports are kept,
30  output ports are discarded. An additional output 'trigger' is created that
31  goes high when an assert is violated. Without a miter_name, the existing
32  module is modified.
33
34      -make_outputs
35          keep module output ports.
36
37      -flatten
38          call 'flatten; opt_expr -keepdc -undriven;;' on the miter circuit.
```

C.76   muxcover – cover trees of MUX cells with wider MUXes

```
1       muxcover [options] [selection]
2
3   Cover trees of $_MUX_ cells with $_MUX{4,8,16}_ cells
4
5       -mux4, -mux8, -mux16
6           Use the specified types of MUXes. If none of those options are used,
7           the effect is the same as if all of them where used.
8
9       -nodecode
10          Do not insert decoder logic. This reduces the number of possible
11          substitutions, but guarantees that the resulting circuit is not
12          less efficient than the original circuit.
```

C.77   nlutmap – map to LUTs of different sizes

```
1       nlutmap [options] [selection]
2
3   This pass uses successive calls to 'abc' to map to an architecture. That
4   provides a small number of differently sized LUTs.
5
6       -luts N_1,N_2,N_3,...
7           The number of LUTs with 1, 2, 3, ... inputs that are
8           available in the target architecture.
9
10      -assert
11          Create an error if not all logic can be mapped
```

```
12
13  Excess logic that does not fit into the specified LUTs is mapped back
14  to generic logic gates ($_AND_, etc.).
```

C.78   opt – perform simple optimizations

```
1       opt [options] [selection]
2
3   This pass calls all the other opt_* passes in a useful order. This performs
4   a series of trivial optimizations and cleanups. This pass executes the other
5   passes in the following order:
6
7       opt_expr [-mux_undef] [-mux_bool] [-undriven] [-clkinv] [-fine] [-full] [-keepdc]
8       opt_merge [-share_all] -nomux
9
10      do
11          opt_muxtree
12          opt_reduce [-fine] [-full]
13          opt_merge [-share_all]
14          opt_rmdff [-keepdc]
15          opt_clean [-purge]
16          opt_expr [-mux_undef] [-mux_bool] [-undriven] [-clkinv] [-fine] [-full] [-keep
17      while <changed design>
18
19  When called with -fast the following script is used instead:
20
21      do
22          opt_expr [-mux_undef] [-mux_bool] [-undriven] [-clkinv] [-fine] [-full] [-keep
23          opt_merge [-share_all]
24          opt_rmdff [-keepdc]
25          opt_clean [-purge]
26      while <changed design in opt_rmdff>
27
28  Note: Options in square brackets (such as [-keepdc]) are passed through to
29  the opt_* commands when given to 'opt'.
```

C.79   opt_clean – remove unused cells and wires

```
1       opt_clean [options] [selection]
2
3   This pass identifies wires and cells that are unused and removes them. Other
4   passes often remove cells but leave the wires in the design or reconnect the
5   wires but leave the old cells in the design. This pass can be used to clean up
6   after the passes that do the actual work.
7
8   This pass only operates on completely selected modules without processes.
9
10      -purge
11          also remove internal nets if they have a public name
```

C.80 opt_expr – perform const folding and simple expression rewriting

```
 1      opt_expr [options] [selection]
 2
 3  This pass performs const folding on internal cell types with constant inputs.
 4  It also performs some simple expression rewritring.
 5
 6      -mux_undef
 7          remove 'undef' inputs from $mux, $pmux and $_MUX_ cells
 8
 9      -mux_bool
10          replace $mux cells with inverters or buffers when possible
11
12      -undriven
13          replace undriven nets with undef (x) constants
14
15      -clkinv
16          optimize clock inverters by changing FF types
17
18      -fine
19          perform fine-grain optimizations
20
21      -full
22          alias for -mux_undef -mux_bool -undriven -fine
23
24      -keepdc
25          some optimizations change the behavior of the circuit with respect to
26          don't-care bits. for example in 'a+0' a single x-bit in 'a' will cause
27          all result bits to be set to x. this behavior changes when 'a+0' is
28          replaced by 'a'. the -keepdc option disables all such optimizations.
```

C.81 opt_merge – consolidate identical cells

```
 1      opt_merge [options] [selection]
 2
 3  This pass identifies cells with identical type and input signals. Such cells
 4  are then merged to one cell.
 5
 6      -nomux
 7          Do not merge MUX cells.
 8
 9      -share_all
10          Operate on all cell types, not just built-in types.
```

C.82 opt_muxtree – eliminate dead trees in multiplexer trees

```
 1      opt_muxtree [selection]
 2
```

```
3  This pass analyzes the control signals for the multiplexer trees in the design
4  and identifies inputs that can never be active. It then removes this dead
5  branches from the multiplexer trees.
6
7  This pass only operates on completely selected modules without processes.
```

C.83 opt_reduce – simplify large MUXes and AND/OR gates

```
1      opt_reduce [options] [selection]
2
3  This pass performs two interlinked optimizations:
4
5  1. it consolidates trees of large AND gates or OR gates and eliminates
6  duplicated inputs.
7
8  2. it identifies duplicated inputs to MUXes and replaces them with a single
9  input with the original control signals OR'ed together.
10
11     -fine
12       perform fine-grain optimizations
13
14     -full
15       alias for -fine
```

C.84 opt_rmdff – remove DFFs with constant inputs

```
1      opt_rmdff [-keepdc] [selection]
2
3  This pass identifies flip-flops with constant inputs and replaces them with
4  a constant driver.
```

C.85 plugin – load and list loaded plugins

```
1      plugin [options]
2
3  Load and list loaded plugins.
4
5     -i <plugin_filename>
6         Load (install) the specified plugin.
7
8     -a <alias_name>
9         Register the specified alias name for the loaded plugin
10
11     -l
12         List loaded plugins
```

C.86   pmuxtree – transform $pmux cells to trees of $mux cells

```
1      pmuxtree [options] [selection]
2
3  This pass transforms $pmux cells to a trees of $mux cells.
```

C.87   prep – generic synthesis script

```
1      prep [options]
2
3  This command runs a conservative RTL synthesis. A typical application for this
4  is the preparation stage of a verification flow. This command does not operate
5  on partly selected designs.
6
7      -top <module>
8          use the specified module as top module (default='top')
9
10     -auto-top
11         automatically determine the top of the design hierarchy
12
13     -flatten
14         flatten the design before synthesis. this will pass '-auto-top' to
15         'hierarchy' if no top module is specified.
16
17     -ifx
18         passed to 'proc'. uses verilog simulation behavior for verilog if/case
19         undef handling. this also prevents 'wreduce' from being run.
20
21     -memx
22         simulate verilog simulation behavior for out-of-bounds memory accesses
23         using the 'memory_memx' pass. This option implies -nordff.
24
25     -nomem
26         do not run any of the memory_* passes
27
28     -nordff
29         passed to 'memory_dff'. prohibits merging of FFs into memory read ports
30
31      -nokeepdc
32         do not call opt_* with -keepdc
33
34     -run <from_label>[:<to_label>]
35         only run the commands between the labels (see below). an empty
36         from label is synonymous to 'begin', and empty to label is
37         synonymous to the end of the command list.
38
39
40  The following commands are executed by this synthesis command:
41
42     begin:
43         hierarchy -check [-top <top> | -auto-top]
44
```

```
45        coarse:
46            proc [-ifx]
47            flatten     (if -flatten)
48            opt_expr -keepdc
49            opt_clean
50            check
51            opt -keepdc
52            wreduce [-memx]
53            memory_dff [-nordff]
54            memory_memx     (if -memx)
55            opt_clean
56            memory_collect
57            opt -keepdc -fast
58
59        check:
60            stat
61            check
```

C.88   proc – translate processes to netlists

```
1        proc [options] [selection]
2
3   This pass calls all the other proc_* passes in the most common order.
4
5        proc_clean
6        proc_rmdead
7        proc_init
8        proc_arst
9        proc_mux
10       proc_dlatch
11       proc_dff
12       proc_clean
13
14   This replaces the processes in the design with multiplexers,
15   flip-flops and latches.
16
17   The following options are supported:
18
19       -global_arst [!]<netname>
20           This option is passed through to proc_arst.
21
22       -ifx
23           This option is passed through to proc_mux. proc_rmdead is not
24           executed in -ifx mode.
```

C.89   proc_arst – detect asynchronous resets

```
1        proc_arst [-global_arst [!]<netname>] [selection]
2
3   This pass identifies asynchronous resets in the processes and converts them
```

```
 4  to a different internal representation that is suitable for generating
 5  flip-flop cells with asynchronous resets.
 6
 7      -global_arst [!]<netname>
 8          In modules that have a net with the given name, use this net as async
 9          reset for registers that have been assign initial values in their
10          declaration ('reg foobar = constant_value;'). Use the '!' modifier for
11          active low reset signals. Note: the frontend stores the default value
12          in the 'init' attribute on the net.
```

C.90   proc_clean – remove empty parts of processes

```
 1      proc_clean [selection]
 2
 3  This pass removes empty parts of processes and ultimately removes a process
 4  if it contains only empty structures.
```

C.91   proc_dff – extract flip-flops from processes

```
 1      proc_dff [selection]
 2
 3  This pass identifies flip-flops in the processes and converts them to
 4  d-type flip-flop cells.
```

C.92   proc_dlatch – extract latches from processes

```
 1      proc_dlatch [selection]
 2
 3  This pass identifies latches in the processes and converts them to
 4  d-type latches.
```

C.93   proc_init – convert initial block to init attributes

```
 1      proc_init [selection]
 2
 3  This pass extracts the 'init' actions from processes (generated from Verilog
 4  'initial' blocks) and sets the initial value to the 'init' attribute on the
 5  respective wire.
```

C.94   proc_mux – convert decision trees to multiplexers

```
 1      proc_mux [options] [selection]
 2
 3  This pass converts the decision trees in processes (originating from if-else
 4  and case statements) to trees of multiplexer cells.
 5
 6      -ifx
 7          Use Verilog simulation behavior with respect to undef values in
 8          'case' expressions and 'if' conditions.
```

C.95   proc_rmdead – eliminate dead trees in decision trees

```
 1      proc_rmdead [selection]
 2
 3  This pass identifies unreachable branches in decision trees and removes them.
```

C.96   qwp – quadratic wirelength placer

```
 1      qwp [options] [selection]
 2
 3  This command runs quadratic wirelength placement on the selected modules and
 4  annotates the cells in the design with 'qwp_position' attributes.
 5
 6      -ltr
 7          Add left-to-right constraints: constrain all inputs on the left border
 8          outputs to the right border.
 9
10      -alpha
11          Add constraints for inputs/outputs to be placed in alphanumerical
12          order along the y-axis (top-to-bottom).
13
14      -grid N
15          Number of grid divisions in x- and y-direction. (default=16)
16
17      -dump <html_file_name>
18          Dump a protocol of the placement algorithm to the html file.
19
20      -v
21          Verbose solver output for profiling or debugging
22
23  Note: This implementation of a quadratic wirelength placer uses exact
24  dense matrix operations. It is only a toy-placer for small circuits.
```

C.97   read_blif – read BLIF file

```
1       read_blif [filename]
2
3   Load modules from a BLIF file into the current design.
4
5       -sop
6           Create $sop cells instead of $lut cells
```

C.98   read_ilang – read modules from ilang file

```
1       read_ilang [filename]
2
3   Load modules from an ilang file to the current design. (ilang is a text
4   representation of a design in yosys's internal format.)
```

C.99   read_liberty – read cells from liberty file

```
1       read_liberty [filename]
2
3   Read cells from liberty file as modules into current design.
4
5       -lib
6           only create empty blackbox modules
7
8       -ignore_redef
9           ignore re-definitions of modules. (the default behavior is to
10          create an error message.)
11
12      -ignore_miss_func
13          ignore cells with missing function specification of outputs
14
15      -ignore_miss_dir
16          ignore cells with a missing or invalid direction
17          specification on a pin
18
19      -setattr <attribute_name>
20          set the specified attribute (to the value 1) on all loaded modules
```

C.100   read_verilog – read modules from Verilog file

```
1       read_verilog [options] [filename]
2
3   Load modules from a Verilog file to the current design. A large subset of
4   Verilog-2005 is supported.
5
6       -sv
7           enable support for SystemVerilog features. (only a small subset
```

```
 8          of SystemVerilog is supported)
 9
10      -formal
11          enable support for SystemVerilog assertions and some Yosys extensions
12          replace the implicit -D SYNTHESIS with -D FORMAL
13
14      -norestrict
15          ignore restrict() assertions
16
17      -assume-asserts
18          treat all assert() statements like assume() statements
19
20      -dump_ast1
21          dump abstract syntax tree (before simplification)
22
23      -dump_ast2
24          dump abstract syntax tree (after simplification)
25
26      -dump_vlog
27          dump ast as Verilog code (after simplification)
28
29      -dump_rtlil
30          dump generated RTLIL netlist
31
32      -yydebug
33          enable parser debug output
34
35      -nolatches
36          usually latches are synthesized into logic loops
37          this option prohibits this and sets the output to 'x'
38          in what would be the latches hold condition
39
40          this behavior can also be achieved by setting the
41          'nolatches' attribute on the respective module or
42          always block.
43
44      -nomem2reg
45          under certain conditions memories are converted to registers
46          early during simplification to ensure correct handling of
47          complex corner cases. this option disables this behavior.
48
49          this can also be achieved by setting the 'nomem2reg'
50          attribute on the respective module or register.
51
52          This is potentially dangerous. Usually the front-end has good
53          reasons for converting an array to a list of registers.
54          Prohibiting this step will likely result in incorrect synthesis
55          results.
56
57      -mem2reg
58          always convert memories to registers. this can also be
59          achieved by setting the 'mem2reg' attribute on the respective
60          module or register.
61
```

```
62        -nomeminit
63            do not infer $meminit cells and instead convert initialized
64            memories to registers directly in the front-end.
65
66        -ppdump
67            dump Verilog code after pre-processor
68
69        -nopp
70            do not run the pre-processor
71
72        -nodpi
73            disable DPI-C support
74
75        -lib
76            only create empty blackbox modules. This implies -DBLACKBOX.
77
78        -noopt
79            don't perform basic optimizations (such as const folding) in the
80            high-level front-end.
81
82        -icells
83            interpret cell types starting with '$' as internal cell types
84
85        -ignore_redef
86            ignore re-definitions of modules. (the default behavior is to
87            create an error message.)
88
89        -defer
90            only read the abstract syntax tree and defer actual compilation
91            to a later 'hierarchy' command. Useful in cases where the default
92            parameters of modules yield invalid or not synthesizable code.
93
94        -noautowire
95            make the default of 'default_nettype be "none" instead of "wire".
96
97        -setattr <attribute_name>
98            set the specified attribute (to the value 1) on all loaded modules
99
100       -Dname[=definition]
101           define the preprocessor symbol 'name' and set its optional value
102           'definition'
103
104       -Idir
105           add 'dir' to the directories which are used when searching include
106           files
107
108   The command 'verilog_defaults' can be used to register default options for
109   subsequent calls to 'read_verilog'.
110
111   Note that the Verilog frontend does a pretty good job of processing valid
112   verilog input, but has not very good error reporting. It generally is
113   recommended to use a simulator (for example Icarus Verilog) for checking
114   the syntax of the code, rather than to rely on read_verilog for that.
115
```

```
116   See the Yosys README file for a list of non-standard Verilog features
117   supported by the Yosys Verilog front-end.
```

C.101   rename – rename object in the design

```
 1        rename old_name new_name
 2
 3    Rename the specified object. Note that selection patterns are not supported
 4    by this command.
 5
 6
 7        rename -enumerate [-pattern <pattern>] [selection]
 8
 9    Assign short auto-generated names to all selected wires and cells with private
10    names. The -pattern option can be used to set the pattern for the new names.
11    The character % in the pattern is replaced with a integer number. The default
12    pattern is '_%_'.
13
14        rename -hide [selection]
15
16    Assign private names (the ones with $-prefix) to all selected wires and cells
17    with public names. This ignores all selected ports.
18
19        rename -top new_name
20
21    Rename top module.
```

C.102   sat – solve a SAT problem in the circuit

```
 1        sat [options] [selection]
 2
 3    This command solves a SAT problem defined over the currently selected circuit
 4    and additional constraints passed as parameters.
 5
 6        -all
 7            show all solutions to the problem (this can grow exponentially, use
 8            -max <N> instead to get <N> solutions)
 9
10        -max <N>
11            like -all, but limit number of solutions to <N>
12
13        -enable_undef
14            enable modeling of undef value (aka 'x-bits')
15            this option is implied by -set-def, -set-undef et. cetera
16
17        -max_undef
18            maximize the number of undef bits in solutions, giving a better
19            picture of which input bits are actually vital to the solution.
20
21        -set <signal> <value>
```

```
22              set the specified signal to the specified value.
23
24          -set-def <signal>
25              add a constraint that all bits of the given signal must be defined
26
27          -set-any-undef <signal>
28              add a constraint that at least one bit of the given signal is undefined
29
30          -set-all-undef <signal>
31              add a constraint that all bits of the given signal are undefined
32
33          -set-def-inputs
34              add -set-def constraints for all module inputs
35
36          -show <signal>
37              show the model for the specified signal. if no -show option is
38              passed then a set of signals to be shown is automatically selected.
39
40          -show-inputs, -show-outputs, -show-ports
41              add all module (input/output) ports to the list of shown signals
42
43          -show-regs, -show-public, -show-all
44              show all registers, show signals with 'public' names, show all signals
45
46          -ignore_div_by_zero
47              ignore all solutions that involve a division by zero
48
49          -ignore_unknown_cells
50              ignore all cells that can not be matched to a SAT model
51
52  The following options can be used to set up a sequential problem:
53
54          -seq <N>
55              set up a sequential problem with <N> time steps. The steps will
56              be numbered from 1 to N.
57
58              note: for large <N> it can be significantly faster to use
59              -tempinduct-baseonly -maxsteps <N> instead of -seq <N>.
60
61          -set-at <N> <signal> <value>
62          -unset-at <N> <signal>
63              set or unset the specified signal to the specified value in the
64              given timestep. this has priority over a -set for the same signal.
65
66          -set-assumes
67              set all assumptions provided via $assume cells
68
69          -set-def-at <N> <signal>
70          -set-any-undef-at <N> <signal>
71          -set-all-undef-at <N> <signal>
72              add undef constraints in the given timestep.
73
74          -set-init <signal> <value>
75              set the initial value for the register driving the signal to the value
```

```
 76
 77        -set-init-undef
 78            set all initial states (not set using -set-init) to undef
 79
 80        -set-init-def
 81            do not force a value for the initial state but do not allow undef
 82
 83        -set-init-zero
 84            set all initial states (not set using -set-init) to zero
 85
 86        -dump_vcd <vcd-file-name>
 87            dump SAT model (counter example in proof) to VCD file
 88
 89        -dump_json <json-file-name>
 90            dump SAT model (counter example in proof) to a WaveJSON file.
 91
 92        -dump_cnf <cnf-file-name>
 93            dump CNF of SAT problem (in DIMACS format). in temporal induction
 94            proofs this is the CNF of the first induction step.
 95
 96    The following additional options can be used to set up a proof. If also -seq
 97    is passed, a temporal induction proof is performed.
 98
 99        -tempinduct
100            Perform a temporal induction proof. In a temporal induction proof it is
101            proven that the condition holds forever after the number of time steps
102            specified using -seq.
103
104        -tempinduct-def
105            Perform a temporal induction proof. Assume an initial state with all
106            registers set to defined values for the induction step.
107
108        -tempinduct-baseonly
109            Run only the basecase half of temporal induction (requires -maxsteps)
110
111        -tempinduct-inductonly
112            Run only the induction half of temporal induction
113
114        -tempinduct-skip <N>
115            Skip the first <N> steps of the induction proof.
116
117            note: this will assume that the base case holds for <N> steps.
118            this must be proven independently with "-tempinduct-baseonly
119            -maxsteps <N>". Use -initsteps if you just want to set a
120            minimal induction length.
121
122        -prove <signal> <value>
123            Attempt to proof that <signal> is always <value>.
124
125        -prove-x <signal> <value>
126            Like -prove, but an undef (x) bit in the lhs matches any value on
127            the right hand side. Useful for equivalence checking.
128
129        -prove-asserts
```

```
130          Prove that all asserts in the design hold.
131
132      -prove-skip <N>
133          Do not enforce the prove-condition for the first <N> time steps.
134
135      -maxsteps <N>
136          Set a maximum length for the induction.
137
138      -initsteps <N>
139          Set initial length for the induction.
140          This will speed up the search of the right induction length
141          for deep induction proofs.
142
143      -stepsize <N>
144          Increase the size of the induction proof in steps of <N>.
145          This will speed up the search of the right induction length
146          for deep induction proofs.
147
148      -timeout <N>
149          Maximum number of seconds a single SAT instance may take.
150
151      -verify
152          Return an error and stop the synthesis script if the proof fails.
153
154      -verify-no-timeout
155          Like -verify but do not return an error for timeouts.
156
157      -falsify
158          Return an error and stop the synthesis script if the proof succeeds.
159
160      -falsify-no-timeout
161          Like -falsify but do not return an error for timeouts.
```

C.103   scatter – add additional intermediate nets

```
1      scatter [selection]
2
3  This command adds additional intermediate nets on all cell ports. This is used
4  for testing the correct use of the SigMap helper in passes. If you don't know
5  what this means: don't worry -- you only need this pass when testing your own
6  extensions to Yosys.
7
8  Use the opt_clean command to get rid of the additional nets.
```

C.104   scc – detect strongly connected components (logic loops)

```
1      scc [options] [selection]
2
3  This command identifies strongly connected components (aka logic loops) in the
4  design.
```

```
 5
 6       -expect <num>
 7           expect to find exactly <num> SSCs. A different number of SSCs will
 8           produce an error.
 9
10       -max_depth <num>
11           limit to loops not longer than the specified number of cells. This
12           can e.g. be useful in identifying small local loops in a module that
13           implements one large SCC.
14
15       -nofeedback
16           do not count cells that have their output fed back into one of their
17           inputs as single-cell scc.
18
19       -all_cell_types
20           Usually this command only considers internal non-memory cells. With
21           this option set, all cells are considered. For unknown cells all ports
22           are assumed to be bidirectional 'inout' ports.
23
24       -set_attr <name> <value>
25       -set_cell_attr <name> <value>
26       -set_wire_attr <name> <value>
27           set the specified attribute on all cells and/or wires that are part of
28           a logic loop. the special token {} in the value is replaced with a
29           unique identifier for the logic loop.
30
31       -select
32           replace the current selection with a selection of all cells and wires
33           that are part of a found logic loop
```

C.105   script – execute commands from script file

```
 1       script <filename> [<from_label>:<to_label>]
 2
 3  This command executes the yosys commands in the specified file.
 4
 5  The 2nd argument can be used to only execute the section of the
 6  file between the specified labels. An empty from label is synonymous
 7  for the beginning of the file and an empty to label is synonymous
 8  for the end of the file.
 9
10  If only one label is specified (without ':') then only the block
11  marked with that label (until the next label) is executed.
```

C.106   select – modify and view the list of selected objects

```
 1       select [ -add | -del | -set <name> ] {-read <filename> | <selection>}
 2       select [ <assert_option> ] {-read <filename> | <selection>}
 3       select [ -list | -write <filename> | -count | -clear ]
 4       select -module <modname>
```

Most commands use the list of currently selected objects to determine which part
of the design to operate on. This command can be used to modify and view this
list of selected objects.

Note that many commands support an optional [selection] argument that can be
used to override the global selection for the command. The syntax of this
optional argument is identical to the syntax of the <selection> argument
described here.

    -add, -del
        add or remove the given objects to the current selection.
        without this options the current selection is replaced.

    -set <name>
        do not modify the current selection. instead save the new selection
        under the given name (see @<name> below). to save the current selection,
        use "select -set <name> %"

    -assert-none
        do not modify the current selection. instead assert that the given
        selection is empty. i.e. produce an error if any object matching the
        selection is found.

    -assert-any
        do not modify the current selection. instead assert that the given
        selection is non-empty. i.e. produce an error if no object matching
        the selection is found.

    -assert-count N
        do not modify the current selection. instead assert that the given
        selection contains exactly N objects.

    -assert-max N
        do not modify the current selection. instead assert that the given
        selection contains less than or exactly N objects.

    -assert-min N
        do not modify the current selection. instead assert that the given
        selection contains at least N objects.

    -list
        list all objects in the current selection

    -write <filename>
        like -list but write the output to the specified file

    -read <filename>
        read the specified file (written by -write)

    -count
        count all objects in the current selection

    -clear

```
 59            clear the current selection. this effectively selects the whole
 60            design. it also resets the selected module (see -module). use the
 61            command 'select *' to select everything but stay in the current module.
 62
 63        -none
 64            create an empty selection. the current module is unchanged.
 65
 66        -module <modname>
 67            limit the current scope to the specified module.
 68            the difference between this and simply selecting the module
 69            is that all object names are interpreted relative to this
 70            module after this command until the selection is cleared again.
 71
 72  When this command is called without an argument, the current selection
 73  is displayed in a compact form (i.e. only the module name when a whole module
 74  is selected).
 75
 76  The <selection> argument itself is a series of commands for a simple stack
 77  machine. Each element on the stack represents a set of selected objects.
 78  After this commands have been executed, the union of all remaining sets
 79  on the stack is computed and used as selection for the command.
 80
 81  Pushing (selecting) object when not in -module mode:
 82
 83        <mod_pattern>
 84            select the specified module(s)
 85
 86        <mod_pattern>/<obj_pattern>
 87            select the specified object(s) from the module(s)
 88
 89  Pushing (selecting) object when in -module mode:
 90
 91        <obj_pattern>
 92            select the specified object(s) from the current module
 93
 94  A <mod_pattern> can be a module name, wildcard expression (*, ?, [..])
 95  matching module names, or one of the following:
 96
 97        A:<pattern>, A:<pattern>=<pattern>
 98            all modules with an attribute matching the given pattern
 99            in addition to = also <, <=, >=, and > are supported
100
101  An <obj_pattern> can be an object name, wildcard expression, or one of
102  the following:
103
104        w:<pattern>
105            all wires with a name matching the given wildcard pattern
106
107        i:<pattern>, o:<pattern>, x:<pattern>
108            all inputs (i:), outputs (o:) or any ports (x:) with matching names
109
110        s:<size>, s:<min>:<max>
111            all wires with a matching width
112
```

```
m:<pattern>
    all memories with a name matching the given pattern

c:<pattern>
    all cells with a name matching the given pattern

t:<pattern>
    all cells with a type matching the given pattern

p:<pattern>
    all processes with a name matching the given pattern

a:<pattern>
    all objects with an attribute name matching the given pattern

a:<pattern>=<pattern>
    all objects with a matching attribute name-value-pair.
    in addition to = also <, <=, >=, and > are supported

r:<pattern>, r:<pattern>=<pattern>
    cells with matching parameters. also with <, <=, >= and >.

n:<pattern>
    all objects with a name matching the given pattern
    (i.e. 'n:' is optional as it is the default matching rule)

@<name>
    push the selection saved prior with 'select -set <name> ...'
```

The following actions can be performed on the top sets on the stack:

```
%
    push a copy of the current selection to the stack

%%
    replace the stack with a union of all elements on it

%n
    replace top set with its invert

%u
    replace the two top sets on the stack with their union

%i
    replace the two top sets on the stack with their intersection

%d
    pop the top set from the stack and subtract it from the new top

%D
    like %d but swap the roles of two top sets on the stack

%c
    create a copy of the top set from the stack and push it
```

```
167
168    %x[<num1>|*][.<num2>][:<rule>[:<rule>..]]
169        expand top set <num1> num times according to the specified rules.
170        (i.e. select all cells connected to selected wires and select all
171        wires connected to selected cells) The rules specify which cell
172        ports to use for this. the syntax for a rule is a '-' for exclusion
173        and a '+' for inclusion, followed by an optional comma separated
174        list of cell types followed by an optional comma separated list of
175        cell ports in square brackets. a rule can also be just a cell or wire
176        name that limits the expansion (is included but does not go beyond).
177        select at most <num2> objects. a warning message is printed when this
178        limit is reached. When '*' is used instead of <num1> then the process
179        is repeated until no further object are selected.
180
181    %ci[<num1>|*][.<num2>][:<rule>[:<rule>..]]
182    %co[<num1>|*][.<num2>][:<rule>[:<rule>..]]
183        similar to %x, but only select input (%ci) or output cones (%co)
184
185    %xe[...] %cie[...] %coe
186        like %x, %ci, and %co but only consider combinatorial cells
187
188    %a
189        expand top set by selecting all wires that are (at least in part)
190        aliases for selected wires.
191
192    %s
193        expand top set by adding all modules that implement cells in selected
194        modules
195
196    %m
197        expand top set by selecting all modules that contain selected objects
198
199    %M
200        select modules that implement selected cells
201
202    %C
203        select cells that implement selected modules
204
205    %R[<num>]
206        select <num> random objects from top selection (default 1)
207
208 Example: the following command selects all wires that are connected to a
209 'GATE' input of a 'SWITCH' cell:
210
211    select */t:SWITCH %x:+[GATE] */t:SWITCH %d
```

C.107   setattr – set/unset attributes on objects

```
1      setattr [ -mod ] [ -set name value | -unset name ]... [selection]
2
3  Set/unset the given attributes on the selected objects. String values must be
4  passed in double quotes (").
```

```
5
6   When called with -mod, this command will set and unset attributes on modules
7   instead of objects within modules.
```

C.108   setparam – set/unset parameters on objects

```
1       setparam [ -type cell_type ] [ -set name value | -unset name ]... [selection]
2
3   Set/unset the given parameters on the selected cells. String values must be
4   passed in double quotes (").
5
6   The -type option can be used to change the cell type of the selected cells.
```

C.109   setundef – replace undef values with defined constants

```
1        setundef [options] [selection]
2
3   This command replaced undef (x) constants with defined (0/1) constants.
4
5       -undriven
6           also set undriven nets to constant values
7
8       -zero
9           replace with bits cleared (0)
10
11      -one
12          replace with bits set (1)
13
14      -random <seed>
15          replace with random bits using the specified integer als seed
16          value for the random number generator.
17
18      -init
19          also create/update init values for flip-flops
```

C.110   share – perform sat-based resource sharing

```
1        share [options] [selection]
2
3   This pass merges shareable resources into a single resource. A SAT solver
4   is used to determine if two resources are share-able.
5
6     -force
7       Per default the selection of cells that is considered for sharing is
8       narrowed using a list of cell types. With this option all selected
9       cells are considered for resource sharing.
10
```

```
11         IMPORTANT NOTE: If the -all option is used then no cells with internal
12         state must be selected!
13
14     -aggressive
15       Per default some heuristics are used to reduce the number of cells
16       considered for resource sharing to only large resources. This options
17       turns this heuristics off, resulting in much more cells being considered
18       for resource sharing.
19
20     -fast
21       Only consider the simple part of the control logic in SAT solving, resulting
22       in much easier SAT problems at the cost of maybe missing some opportunities
23       for resource sharing.
24
25     -limit N
26       Only perform the first N merges, then stop. This is useful for debugging.
```

C.111 shell – enter interactive command mode

```
1       shell
2
3  This command enters the interactive command mode. This can be useful
4  in a script to interrupt the script at a certain point and allow for
5  interactive inspection or manual synthesis of the design at this point.
6
7  The command prompt of the interactive shell indicates the current
8  selection (see 'help select'):
9
10     yosys>
11         the entire design is selected
12
13     yosys*>
14         only part of the design is selected
15
16     yosys [modname]>
17         the entire module 'modname' is selected using 'select -module modname'
18
19     yosys [modname]*>
20         only part of current module 'modname' is selected
21
22  When in interactive shell, some errors (e.g. invalid command arguments)
23  do not terminate yosys but return to the command prompt.
24
25  This command is the default action if nothing else has been specified
26  on the command line.
27
28  Press Ctrl-D or type 'exit' to leave the interactive shell.
```

C.112 show – generate schematics using graphviz

```
show [options] [selection]

Create a graphviz DOT file for the selected part of the design and compile it
to a graphics file (usually SVG or PostScript).

    -viewer <viewer>
        Run the specified command with the graphics file as parameter.

    -format <format>
        Generate a graphics file in the specified format. Use 'dot' to just
        generate a .dot file, or other <format> strings such as 'svg' or 'ps'
        to generate files in other formats (this calls the 'dot' command).

    -lib <verilog_or_ilang_file>
        Use the specified library file for determining whether cell ports are
        inputs or outputs. This option can be used multiple times to specify
        more than one library.

        note: in most cases it is better to load the library before calling
        show with 'read_verilog -lib <filename>'. it is also possible to
        load liberty files with 'read_liberty -lib <filename>'.

    -prefix <prefix>
        generate <prefix>.* instead of ~/.yosys_show.*

    -color <color> <object>
        assign the specified color to the specified object. The object can be
        a single selection wildcard expressions or a saved set of objects in
        the @<name> syntax (see "help select" for details).

    -label <text> <object>
        assign the specified label text to the specified object. The object can
        be a single selection wildcard expressions or a saved set of objects in
        the @<name> syntax (see "help select" for details).

    -colors <seed>
        Randomly assign colors to the wires. The integer argument is the seed
        for the random number generator. Change the seed value if the colored
        graph still is ambiguous. A seed of zero deactivates the coloring.

    -colorattr <attribute_name>
        Use the specified attribute to assign colors. A unique color is
        assigned to each unique value of this attribute.

    -width
        annotate busses with a label indicating the width of the bus.

    -signed
        mark ports (A, B) that are declared as signed (using the [AB]_SIGNED
        cell parameter) with an asterisk next to the port name.

    -stretch
        stretch the graph so all inputs are on the left side and all outputs
        (including inout ports) are on the right side.
```

```
55
56      -pause
57          wait for the use to press enter to before returning
58
59      -enum
60          enumerate objects with internal ($-prefixed) names
61
62      -long
63          do not abbreviate objects with internal ($-prefixed) names
64
65      -notitle
66          do not add the module name as graph title to the dot file
67
68  When no <format> is specified, 'dot' is used. When no <format> and <viewer> is
69  specified, 'xdot' is used to display the schematic.
70
71  The generated output files are '~/.yosys_show.dot' and '~/.yosys_show.<format>',
72  unless another prefix is specified using -prefix <prefix>.
73
74  Yosys on Windows and YosysJS use different defaults: The output is written
75  to 'show.dot' in the current directory and new viewer is launched each time
76  the 'show' command is executed.
```

C.113   shregmap – map shift registers

```
1       shregmap [options] [selection]
2
3   This pass converts chains of $_DFF_[NP]_ gates to target specific shift register
4   primitives. The generated shift register will be of type $__SHREG_DFF_[NP]_ and
5   will use the same interface as the original $_DFF_*_ cells. The cell parameter
6   'DEPTH' will contain the depth of the shift register. Use a target-specific
7   'techmap' map file to convert those cells to the actual target cells.
8
9       -minlen N
10          minimum length of shift register (default = 2)
11          (this is the length after -keep_before and -keep_after)
12
13      -maxlen N
14          maximum length of shift register (default = no limit)
15          larger chains will be mapped to multiple shift register instances
16
17      -keep_before N
18          number of DFFs to keep before the shift register (default = 0)
19
20      -keep_after N
21          number of DFFs to keep after the shift register (default = 0)
22
23      -clkpol pos|neg|any
24          limit match to only positive or negative edge clocks. (default = any)
25
26      -enpol pos|neg|none|any_or_none|any
27          limit match to FFs with the specified enable polarity. (default = none)
```

```
28
29      -match <cell_type>[:<d_port_name>:<q_port_name>]
30          match the specified cells instead of $_DFF_N_ and $_DFF_P_. If
31          ':<d_port_name>:<q_port_name>' is omitted then 'D' and 'Q' is used
32          by default. E.g. the option '-clkpol pos' is just an alias for
33          '-match $_DFF_P_', which is an alias for '-match $_DFF_P_:D:Q'.
34
35      -params
36          instead of encoding the clock and enable polarity in the cell name by
37          deriving from the original cell name, simply name all generated cells
38          $__SHREG_ and use CLKPOL and ENPOL parameters. An ENPOL value of 2 is
39          used to denote cells without enable input. The ENPOL parameter is
40          omitted when '-enpol none' (or no -enpol option) is passed.
41
42      -zinit
43          assume the shift register is automatically zero-initialized, so it
44          becomes legal to merge zero initialized FFs into the shift register.
45
46      -init
47          map initialized registers to the shift reg, add an INIT parameter to
48          generated cells with the initialization value. (first bit to shift out
49          in LSB position)
50
51      -tech greenpak4
52          map to greenpak4 shift registers.
```

C.114   simplemap – mapping simple coarse-grain cells

```
1       simplemap [selection]
2
3   This pass maps a small selection of simple coarse-grain cells to yosys gate
4   primitives. The following internal cell types are mapped by this pass:
5
6     $not, $pos, $and, $or, $xor, $xnor
7     $reduce_and, $reduce_or, $reduce_xor, $reduce_xnor, $reduce_bool
8     $logic_not, $logic_and, $logic_or, $mux, $tribuf
9     $sr, $ff, $dff, $dffsr, $adff, $dlatch
```

C.115   singleton – create singleton modules

```
1       singleton [selection]
2
3   By default, a module that is instantiated by several other modules is only
4   kept once in the design. This preserves the original modularity of the design
5   and reduces the overall size of the design in memory. But it prevents certain
6   optimizations and other operations on the design. This pass creates singleton
7   modules for all selected cells. The created modules are marked with the
8   'singleton' attribute.
9
10  This commands only operates on modules that by themself have the 'singleton'
```

```
11  attribute set (the 'top' module is a singleton implicitly).
```

C.116   splice – create explicit splicing cells

```
 1      splice [options] [selection]
 2
 3  This command adds $slice and $concat cells to the design to make the splicing
 4  of multi-bit signals explicit. This for example is useful for coarse grain
 5  synthesis, where dedicated hardware is needed to splice signals.
 6
 7      -sel_by_cell
 8          only select the cell ports to rewire by the cell. if the selection
 9          contains a cell, than all cell inputs are rewired, if necessary.
10
11      -sel_by_wire
12          only select the cell ports to rewire by the wire. if the selection
13          contains a wire, than all cell ports driven by this wire are wired,
14          if necessary.
15
16      -sel_any_bit
17          it is sufficient if the driver of any bit of a cell port is selected.
18          by default all bits must be selected.
19
20      -wires
21          also add $slice and $concat cells to drive otherwise unused wires.
22
23      -no_outputs
24          do not rewire selected module outputs.
25
26      -port <name>
27          only rewire cell ports with the specified name. can be used multiple
28          times. implies -no_output.
29
30      -no_port <name>
31          do not rewire cell ports with the specified name. can be used multiple
32          times. can not be combined with -port <name>.
33
34  By default selected output wires and all cell ports of selected cells driven
35  by selected wires are rewired.
```

C.117   splitnets – split up multi-bit nets

```
 1      splitnets [options] [selection]
 2
 3  This command splits multi-bit nets into single-bit nets.
 4
 5      -format char1[char2[char3]]
 6          the first char is inserted between the net name and the bit index, the
 7          second char is appended to the netname. e.g. -format () creates net
 8          names like 'mysignal(42)'. the 3rd character is the range separation
```

```
 9          character when creating multi-bit wires. the default is ’[]:’.
10
11      -ports
12          also split module ports. per default only internal signals are split.
13
14      -driver
15          don’t blindly split nets in individual bits. instead look at the driver
16          and split nets so that no driver drives only part of a net.
```

C.118   stat – print some statistics

```
 1      stat [options] [selection]
 2
 3  Print some statistics (number of objects) on the selected portion of the
 4  design.
 5
 6      -top <module>
 7          print design hierarchy with this module as top. if the design is fully
 8          selected and a module has the ’top’ attribute set, this module is used
 9          default value for this option.
10
11      -liberty <liberty_file>
12          use cell area information from the provided liberty file
13
14      -width
15          annotate internal cell types with their word width.
16          e.g. $add_8 for an 8 bit wide $add cell.
```

C.119   submod – moving part of a module to a new submodule

```
 1      submod [-copy] [selection]
 2
 3  This pass identifies all cells with the ’submod’ attribute and moves them to
 4  a newly created module. The value of the attribute is used as name for the
 5  cell that replaces the group of cells with the same attribute value.
 6
 7  This pass can be used to create a design hierarchy in flat design. This can
 8  be useful for analyzing or reverse-engineering a design.
 9
10  This pass only operates on completely selected modules with no processes
11  or memories.
12
13
14      submod -name <name> [-copy] [selection]
15
16  As above, but don’t use the ’submod’ attribute but instead use the selection.
17  Only objects from one module might be selected. The value of the -name option
18  is used as the value of the ’submod’ attribute above.
19
20  By default the cells are ’moved’ from the source module and the source module
```

```
21  will use an instance of the new module after this command is finished. Call
22  with -copy to not modify the source module.
```

C.120   synth – generic synthesis script

```
 1       synth [options]
 2
 3  This command runs the default synthesis script. This command does not operate
 4  on partly selected designs.
 5
 6      -top <module>
 7          use the specified module as top module (default='top')
 8
 9      -auto-top
10          automatically determine the top of the design hierarchy
11
12      -flatten
13          flatten the design before synthesis. this will pass '-auto-top' to
14          'hierarchy' if no top module is specified.
15
16      -encfile <file>
17          passed to 'fsm_recode' via 'fsm'
18
19      -nofsm
20          do not run FSM optimization
21
22      -noabc
23          do not run abc (as if yosys was compiled without ABC support)
24
25      -noalumacc
26          do not run 'alumacc' pass. i.e. keep arithmetic operators in
27          their direct form ($add, $sub, etc.).
28
29      -nordff
30          passed to 'memory'. prohibits merging of FFs into memory read ports
31
32      -run <from_label>[:<to_label>]
33          only run the commands between the labels (see below). an empty
34          from label is synonymous to 'begin', and empty to label is
35          synonymous to the end of the command list.
36
37
38  The following commands are executed by this synthesis command:
39
40      begin:
41          hierarchy -check [-top <top> | -auto-top]
42
43      coarse:
44          proc
45          flatten     (if -flatten)
46          opt_expr
47          opt_clean
```

```
48          check
49          opt
50          wreduce
51          alumacc
52          share
53          opt
54          fsm
55          opt -fast
56          memory -nomap
57          opt_clean
58
59      fine:
60          opt -fast -full
61          memory_map
62          opt -full
63          techmap
64          opt -fast
65          abc -fast
66          opt -fast
67
68      check:
69          hierarchy -check
70          stat
71          check
```

C.121   synth_gowin – synthesis for Gowin FPGAs

```
1       synth_gowin [options]
2
3   This command runs synthesis for Gowin FPGAs. This work is experimental.
4
5       -top <module>
6           use the specified module as top module (default='top')
7
8       -vout <file>
9           write the design to the specified Verilog netlist file. writing of an
10          output file is omitted if this parameter is not specified.
11
12      -run <from_label>:<to_label>
13          only run the commands between the labels (see below). an empty
14          from label is synonymous to 'begin', and empty to label is
15          synonymous to the end of the command list.
16
17      -retime
18          run 'abc' with -dff option
19
20
21  The following commands are executed by this synthesis command:
22
23      begin:
24          read_verilog -lib +/gowin/cells_sim.v
25          hierarchy -check -top <top>
```

```
26
27       flatten:
28           proc
29           flatten
30           tribuf -logic
31           deminout
32
33       coarse:
34           synth -run coarse
35
36       fine:
37           opt -fast -mux_undef -undriven -fine
38           memory_map
39           opt -undriven -fine
40           techmap
41           clean -purge
42           splitnets -ports
43           setundef -undriven -zero
44           abc -dff    (only if -retime)
45
46       map_luts:
47           abc -lut 4
48           clean
49
50       map_cells:
51           techmap -map +/gowin/cells_map.v
52           hilomap -hicell VCC V -locell GND G
53           iopadmap -inpad IBUF O:I -outpad OBUF I:O
54           clean -purge
55
56       check:
57           hierarchy -check
58           stat
59           check -noinit
60
61       vout:
62           write_verilog -attr2comment -defparam -renameprefix gen <file-name>
```

C.122   synth_greenpak4 – synthesis for GreenPAK4 FPGAs

```
1       synth_greenpak4 [options]
2
3  This command runs synthesis for GreenPAK4 FPGAs. This work is experimental.
4
5      -top <module>
6          use the specified module as top module (default='top')
7
8      -part <part>
9          synthesize for the specified part. Valid values are SLG46140V,
10          SLG46620V, and SLG46621V (default).
11
12      -json <file>
```

```
13              write the design to the specified JSON file. writing of an output file
14              is omitted if this parameter is not specified.
15
16      -run <from_label>:<to_label>
17              only run the commands between the labels (see below). an empty
18              from label is synonymous to 'begin', and empty to label is
19              synonymous to the end of the command list.
20
21      -noflatten
22              do not flatten design before synthesis
23
24      -retime
25              run 'abc' with -dff option
26
27
28  The following commands are executed by this synthesis command:
29
30      begin:
31          read_verilog -lib +/greenpak4/cells_sim.v
32          hierarchy -check -top <top>
33
34      flatten:     (unless -noflatten)
35          proc
36          flatten
37          tribuf -logic
38
39      coarse:
40          synth -run coarse
41
42      fine:
43          greenpak4_counters
44          clean
45          opt -fast -mux_undef -undriven -fine
46          memory_map
47          opt -undriven -fine
48          techmap
49          dfflibmap -prepare -liberty +/greenpak4/gp_dff.lib
50          opt -fast
51          abc -dff    (only if -retime)
52
53      map_luts:
54          nlutmap -assert -luts 0,6,8,2      (for -part SLG46140V)
55          nlutmap -assert -luts 2,8,16,2     (for -part SLG46620V)
56          nlutmap -assert -luts 2,8,16,2     (for -part SLG46621V)
57          clean
58
59      map_cells:
60          shregmap -tech greenpak4
61          dfflibmap -liberty +/greenpak4/gp_dff.lib
62          dffinit -ff GP_DFF Q INIT
63          dffinit -ff GP_DFFR Q INIT
64          dffinit -ff GP_DFFS Q INIT
65          dffinit -ff GP_DFFSR Q INIT
66          iopadmap -bits -inpad GP_IBUF OUT:IN -outpad GP_OBUF IN:OUT -inoutpad GP_OBUF
```

```
67            attrmvcp -attr src -attr LOC t:GP_OBUF t:GP_OBUFT t:GP_IOBUF n:*
68            attrmvcp -attr src -attr LOC -driven t:GP_IBUF n:*
69            techmap -map +/greenpak4/cells_map.v
70            greenpak4_dffinv
71            clean
72
73        check:
74            hierarchy -check
75            stat
76            check -noinit
77
78        json:
79            write_json <file-name>
```

C.123 synth_ice40 – synthesis for iCE40 FPGAs

```
1        synth_ice40 [options]
2
3    This command runs synthesis for iCE40 FPGAs.
4
5        -top <module>
6            use the specified module as top module (default='top')
7
8        -blif <file>
9            write the design to the specified BLIF file. writing of an output file
10           is omitted if this parameter is not specified.
11
12       -edif <file>
13           write the design to the specified edif file. writing of an output file
14           is omitted if this parameter is not specified.
15
16       -run <from_label>:<to_label>
17           only run the commands between the labels (see below). an empty
18           from label is synonymous to 'begin', and empty to label is
19           synonymous to the end of the command list.
20
21       -noflatten
22           do not flatten design before synthesis
23
24       -retime
25           run 'abc' with -dff option
26
27       -nocarry
28           do not use SB_CARRY cells in output netlist
29
30       -nobram
31           do not use SB_RAM40_4K* cells in output netlist
32
33       -abc2
34           run two passes of 'abc' for slightly improved logic density
35
36
```

```
37   The following commands are executed by this synthesis command:
38
39       begin:
40           read_verilog -lib +/ice40/cells_sim.v
41           hierarchy -check -top <top>
42
43       flatten:     (unless -noflatten)
44           proc
45           flatten
46           tribuf -logic
47           deminout
48
49       coarse:
50           synth -run coarse
51
52       bram:    (skip if -nobram)
53           memory_bram -rules +/ice40/brams.txt
54           techmap -map +/ice40/brams_map.v
55
56       fine:
57           opt -fast -mux_undef -undriven -fine
58           memory_map
59           opt -undriven -fine
60           techmap -map +/techmap.v -map +/ice40/arith_map.v
61           abc -dff     (only if -retime)
62           ice40_opt
63
64       map_ffs:
65           dffsr2dff
66           dff2dffe -direct-match $_DFF_*
67           techmap -map +/ice40/cells_map.v
68           opt_expr -mux_undef
69           simplemap
70           ice40_ffinit
71           ice40_ffssr
72           ice40_opt -full
73
74       map_luts:
75           abc           (only if -abc2)
76           ice40_opt     (only if -abc2)
77           techmap -map +/ice40/latches_map.v
78           abc -lut 4
79           clean
80
81       map_cells:
82           techmap -map +/ice40/cells_map.v
83           clean
84
85       check:
86           hierarchy -check
87           stat
88           check -noinit
89
90       blif:
```

```
91        write_blif -gates -attr -param <file-name>
92
93    edif:
94        write_edif <file-name>
```

C.124   synth_xilinx – synthesis for Xilinx FPGAs

```
 1      synth_xilinx [options]
 2
 3  This command runs synthesis for Xilinx FPGAs. This command does not operate on
 4  partly selected designs. At the moment this command creates netlists that are
 5  compatible with 7-Series Xilinx devices.
 6
 7      -top <module>
 8          use the specified module as top module
 9
10      -edif <file>
11          write the design to the specified edif file. writing of an output file
12          is omitted if this parameter is not specified.
13
14      -run <from_label>:<to_label>
15          only run the commands between the labels (see below). an empty
16          from label is synonymous to 'begin', and empty to label is
17          synonymous to the end of the command list.
18
19      -flatten
20          flatten design before synthesis
21
22      -retime
23          run 'abc' with -dff option
24
25
26  The following commands are executed by this synthesis command:
27
28      begin:
29          read_verilog -lib +/xilinx/cells_sim.v
30          read_verilog -lib +/xilinx/cells_xtra.v
31          read_verilog -lib +/xilinx/brams_bb.v
32          read_verilog -lib +/xilinx/drams_bb.v
33          hierarchy -check -top <top>
34
35      flatten:      (only if -flatten)
36          proc
37          flatten
38
39      coarse:
40          synth -run coarse
41
42      bram:
43          memory_bram -rules +/xilinx/brams.txt
44          techmap -map +/xilinx/brams_map.v
45
```

```
46      dram:
47          memory_bram -rules +/xilinx/drams.txt
48          techmap -map +/xilinx/drams_map.v
49
50      fine:
51          opt -fast -full
52          memory_map
53          dffsr2dff
54          dff2dffe
55          opt -full
56          techmap -map +/techmap.v -map +/xilinx/arith_map.v
57          opt -fast
58
59      map_luts:
60          abc -luts 2:2,3,6:5,10,20 [-dff]
61          clean
62
63      map_cells:
64          techmap -map +/xilinx/cells_map.v
65          dffinit -ff FDRE Q INIT -ff FDCE Q INIT -ff FDPE Q INIT
66          clean
67
68      check:
69          hierarchy -check
70          stat
71          check -noinit
72
73      edif:      (only if -edif)
74          write_edif <file-name>
```

C.125 tcl – execute a TCL script file

```
1      tcl <filename>
2
3  This command executes the tcl commands in the specified file.
4  Use 'yosys cmd' to run the yosys command 'cmd' from tcl.
5
6  The tcl command 'yosys -import' can be used to import all yosys
7  commands directly as tcl commands to the tcl shell. The yosys
8  command 'proc' is wrapped using the tcl command 'procs' in order
9  to avoid a name collision with the tcl builtin command 'proc'.
```

C.126 techmap – generic technology mapper

```
1      techmap [-map filename] [selection]
2
3  This pass implements a very simple technology mapper that replaces cells in
4  the design with implementations given in form of a Verilog or ilang source
5  file.
6
```

```
 7      -map filename
 8          the library of cell implementations to be used.
 9          without this parameter a builtin library is used that
10          transforms the internal RTL cells to the internal gate
11          library.
12
13      -map %<design-name>
14          like -map above, but with an in-memory design instead of a file.
15
16      -extern
17          load the cell implementations as separate modules into the design
18          instead of inlining them.
19
20      -max_iter <number>
21          only run the specified number of iterations.
22
23      -recursive
24          instead of the iterative breadth-first algorithm use a recursive
25          depth-first algorithm. both methods should yield equivalent results,
26          but may differ in performance.
27
28      -autoproc
29          Automatically call "proc" on implementations that contain processes.
30
31      -assert
32          this option will cause techmap to exit with an error if it can't map
33          a selected cell. only cell types that end on an underscore are accepted
34          as final cell types by this mode.
35
36      -D <define>, -I <incdir>
37          this options are passed as-is to the Verilog frontend for loading the
38          map file. Note that the Verilog frontend is also called with the
39          '-ignore_redef' option set.
40
41  When a module in the map file has the 'techmap_celltype' attribute set, it will
42  match cells with a type that match the text value of this attribute. Otherwise
43  the module name will be used to match the cell.
44
45  When a module in the map file has the 'techmap_simplemap' attribute set, techmap
46  will use 'simplemap' (see 'help simplemap') to map cells matching the module.
47
48  When a module in the map file has the 'techmap_maccmap' attribute set, techmap
49  will use 'maccmap' (see 'help maccmap') to map cells matching the module.
50
51  When a module in the map file has the 'techmap_wrap' attribute set, techmap
52  will create a wrapper for the cell and then run the command string that the
53  attribute is set to on the wrapper module.
54
55  All wires in the modules from the map file matching the pattern _TECHMAP_*
56  or *._TECHMAP_* are special wires that are used to pass instructions from
57  the mapping module to the techmap command. At the moment the following special
58  wires are supported:
59
60      _TECHMAP_FAIL_
```

```
 61          When this wire is set to a non-zero constant value, techmap will not
 62          use this module and instead try the next module with a matching
 63          'techmap_celltype' attribute.
 64
 65          When such a wire exists but does not have a constant value after all
 66          _TECHMAP_DO_* commands have been executed, an error is generated.
 67
 68      _TECHMAP_DO_*
 69          This wires are evaluated in alphabetical order. The constant text value
 70          of this wire is a yosys command (or sequence of commands) that is run
 71          by techmap on the module. A common use case is to run 'proc' on modules
 72          that are written using always-statements.
 73
 74          When such a wire has a non-constant value at the time it is to be
 75          evaluated, an error is produced. That means it is possible for such a
 76          wire to start out as non-constant and evaluate to a constant value
 77          during processing of other _TECHMAP_DO_* commands.
 78
 79          A _TECHMAP_DO_* command may start with the special token 'CONSTMAP; '.
 80          in this case techmap will create a copy for each distinct configuration
 81          of constant inputs and shorted inputs at this point and import the
 82          constant and connected bits into the map module. All further commands
 83          are executed in this copy. This is a very convenient way of creating
 84          optimized specializations of techmap modules without using the special
 85          parameters described below.
 86
 87          A _TECHMAP_DO_* command may start with the special token 'RECURSION; '.
 88          then techmap will recursively replace the cells in the module with their
 89          implementation. This is not affected by the -max_iter option.
 90
 91          It is possible to combine both prefixes to 'RECURSION; CONSTMAP; '.
 92
 93  In addition to this special wires, techmap also supports special parameters in
 94  modules in the map file:
 95
 96      _TECHMAP_CELLTYPE_
 97          When a parameter with this name exists, it will be set to the type name
 98          of the cell that matches the module.
 99
100      _TECHMAP_CONSTMSK_<port-name>_
101      _TECHMAP_CONSTVAL_<port-name>_
102          When this pair of parameters is available in a module for a port, then
103          former has a 1-bit for each constant input bit and the latter has the
104          value for this bit. The unused bits of the latter are set to undef (x).
105
106      _TECHMAP_BITS_CONNMAP_
107      _TECHMAP_CONNMAP_<port-name>_
108          For an N-bit port, the _TECHMAP_CONNMAP_<port-name>_ parameter, if it
109          exists, will be set to an N*_TECHMAP_BITS_CONNMAP_ bit vector containing
110          N words (of _TECHMAP_BITS_CONNMAP_ bits each) that assign each single
111          bit driver a unique id. The values 0-3 are reserved for 0, 1, x, and z.
112          This can be used to detect shorted inputs.
113
114  When a module in the map file has a parameter where the according cell in the
```

```
115  design has a port, the module from the map file is only used if the port in
116  the design is connected to a constant value. The parameter is then set to the
117  constant value.
118
119  A cell with the name _TECHMAP_REPLACE_ in the map file will inherit the name
120  of the cell that is being replaced.
121
122  See 'help extract' for a pass that does the opposite thing.
123
124  See 'help flatten' for a pass that does flatten the design (which is
125  essentially techmap but using the design itself as map library).
```

C.127   tee – redirect command output to file

```
1      tee [-q] [-o logfile|-a logfile] cmd
2
3  Execute the specified command, optionally writing the commands output to the
4  specified logfile(s).
5
6      -q
7          Do not print output to the normal destination (console and/or log file)
8
9      -o logfile
10         Write output to this file, truncate if exists.
11
12     -a logfile
13         Write output to this file, append if exists.
14
15     +INT, -INT
16         Add/subract INT from the -v setting for this command.
```

C.128   test_abcloop – automatically test handling of loops in abc command

```
1      test_abcloop [options]
2
3  Test handling of logic loops in ABC.
4
5      -n {integer}
6          create this number of circuits and test them (default = 100).
7
8      -s {positive_integer}
9          use this value as rng seed value (default = unix time).
```

C.129   test_autotb – generate simple test benches

```
1      test_autotb [options] [filename]
2
3  Automatically create primitive Verilog test benches for all modules in the
4  design. The generated testbenches toggle the input pins of the module in
5  a semi-random manner and dumps the resulting output signals.
6
7  This can be used to check the synthesis results for simple circuits by
8  comparing the testbench output for the input files and the synthesis results.
9
10 The backend automatically detects clock signals. Additionally a signal can
11 be forced to be interpreted as clock signal by setting the attribute
12 'gentb_clock' on the signal.
13
14 The attribute 'gentb_constant' can be used to force a signal to a constant
15 value after initialization. This can e.g. be used to force a reset signal
16 low in order to explore more inner states in a state machine.
17
18     -n <int>
19         number of iterations the test bench should run (default = 1000)
```

C.130 test_cell – automatically test the implementation of a cell type

```
1      test_cell [options] {cell-types}
2
3  Tests the internal implementation of the given cell type (for example '$add')
4  by comparing SAT solver, EVAL and TECHMAP implementations of the cell types..
5
6  Run with 'all' instead of a cell type to run the test on all supported
7  cell types. Use for example 'all /$add' for all cell types except $add.
8
9     -n {integer}
10        create this number of cell instances and test them (default = 100).
11
12    -s {positive_integer}
13        use this value as rng seed value (default = unix time).
14
15    -f {ilang_file}
16        don't generate circuits. instead load the specified ilang file.
17
18    -w {filename_prefix}
19        don't test anything. just generate the circuits and write them
20        to ilang files with the specified prefix
21
22    -map {filename}
23        pass this option to techmap.
24
25    -simlib
26        use "techmap -D SIMLIB_NOCHECKS -map +/simlib.v -max_iter 2 -autoproc"
27
28    -aigmap
29        instead of calling "techmap", call "aigmap"
```

```
30
31     -muxdiv
32         when creating test benches with dividers, create an additional mux
33         to mask out the division-by-zero case
34
35     -script {script_file}
36         instead of calling "techmap", call "script {script_file}".
37
38     -const
39         set some input bits to random constant values
40
41     -nosat
42         do not check SAT model or run SAT equivalence checking
43
44     -noeval
45         do not check const-eval models
46
47     -edges
48         test cell edges db creator against sat-based implementation
49
50     -v
51         print additional debug information to the console
52
53     -vlog {filename}
54         create a Verilog test bench to test simlib and write_verilog
```

C.131   torder – print cells in topological order

```
1      torder [options] [selection]
2
3  This command prints the selected cells in topological order.
4
5      -stop <cell_type> <cell_port>
6          do not use the specified cell port in topological sorting
7
8      -noautostop
9          by default Q outputs of internal FF cells and memory read port outputs
10         are not used in topological sorting. this option deactivates that.
```

C.132   trace – redirect command output to file

```
1      trace cmd
2
3  Execute the specified command, logging all changes the command performs on
4  the design in real time.
```

C.133   tribuf – infer tri-state buffers

```
1      tribuf [options] [selection]
2
3  This pass transforms $mux cells with 'z' inputs to tristate buffers.
4
5      -merge
6          merge multiple tri-state buffers driving the same net
7          into a single buffer.
8
9      -logic
10         convert tri-state buffers that do not drive output ports
11         to non-tristate logic. this option implies -merge.
```

C.134 verific – load Verilog and VHDL designs using Verific

```
1      verific {-vlog95|-vlog2k|-sv2005|-sv2009|-sv} <verilog-file>..
2
3  Load the specified Verilog/SystemVerilog files into Verific.
4
5
6      verific {-vhdl87|-vhdl93|-vhdl2k|-vhdl2008} <vhdl-file>..
7
8  Load the specified VHDL files into Verific.
9
10
11     verific -import [-gates] {-all | <top-module>..}
12
13 Elaborate the design for the specified top modules, import to Yosys and
14 reset the internal state of Verific. A gate-level netlist is created
15 when called with -gates.
16
17 Visit http://verific.com/ for more information on Verific.
```

C.135 verilog_defaults – set default options for read_verilog

```
1      verilog_defaults -add [options]
2
3  Add the specified options to the list of default options to read_verilog.
4
5
6      verilog_defaults -clear
7
8  Clear the list of Verilog default options.
9
10
11     verilog_defaults -push
12     verilog_defaults -pop
13
14 Push or pop the list of default options to a stack. Note that -push does
15 not imply -clear.
```

C.136 vhdl2verilog – importing VHDL designs using vhdl2verilog

```
    vhdl2verilog [options] <vhdl-file>..

This command reads VHDL source files using the 'vhdl2verilog' tool and the
Yosys Verilog frontend.

    -out <out_file>
        do not import the vhdl2verilog output. instead write it to the
        specified file.

    -vhdl2verilog_dir <directory>
        do use the specified vhdl2verilog installation. this is the directory
        that contains the setup_env.sh file. when this option is not present,
        it is assumed that vhdl2verilog is in the PATH environment variable.

    -top <top-entity-name>
        The name of the top entity. This option is mandatory.

The following options are passed as-is to vhdl2verilog:

    -arch <architecture_name>
    -unroll_generate
    -nogenericeval
    -nouniquify
    -oldparser
    -suppress <list>
    -quiet
    -nobanner
    -mapfile <file>

vhdl2verilog can be obtained from:
http://www.edautils.com/vhdl2verilog.html
```

C.137 wreduce – reduce the word size of operations if possible

```
    wreduce [options] [selection]

This command reduces the word size of operations. For example it will replace
the 32 bit adders in the following code with adders of more appropriate widths:

    module test(input [3:0] a, b, c, output [7:0] y);
        assign y = a + b + c + 1;
    endmodule

Options:

    -memx
        Do not change the width of memory address ports. Use this options in
        flows that use the 'memory_memx' pass.
```

C.138   write_blif – write design to BLIF file

```
write_blif [options] [filename]

Write the current design to an BLIF file.

    -top top_module
        set the specified module as design top module

    -buf <cell-type> <in-port> <out-port>
        use cells of type <cell-type> with the specified port names for buffers

    -unbuf <cell-type> <in-port> <out-port>
        replace buffer cells with the specified name and port names with
        a .names statement that models a buffer

    -true <cell-type> <out-port>
    -false <cell-type> <out-port>
    -undef <cell-type> <out-port>
        use the specified cell types to drive nets that are constant 1, 0, or
        undefined. when '-' is used as <cell-type>, then <out-port> specifies
        the wire name to be used for the constant signal and no cell driving
        that wire is generated. when '+' is used as <cell-type>, then <out-port>
        specifies the wire name to be used for the constant signal and a .names
        statement is generated to drive the wire.

    -noalias
        if a net name is aliasing another net name, then by default a net
        without fanout is created that is driven by the other net. This option
        suppresses the generation of this nets without fanout.

The following options can be useful when the generated file is not going to be
read by a BLIF parser but a custom tool. It is recommended to not name the output
file *.blif when any of this options is used.

    -icells
        do not translate Yosys's internal gates to generic BLIF logic
        functions. Instead create .subckt or .gate lines for all cells.

    -gates
        print .gate instead of .subckt lines for all cells that are not
        instantiations of other modules from this design.

    -conn
        do not generate buffers for connected wires. instead use the
        non-standard .conn statement.

    -attr
        use the non-standard .attr statement to write cell attributes

    -param
        use the non-standard .param statement to write cell parameters

    -cname
```

```
53          use the non-standard .cname statement to write cell names
54
55      -blackbox
56          write blackbox cells with .blackbox statement.
57
58      -impltf
59          do not write definitions for the $true, $false and $undef wires.
```

C.139   write_btor – write design to BTOR file

```
1      write_btor [filename]
2
3  Write the current design to an BTOR file.
```

C.140   write_edif – write design to EDIF netlist file

```
1      write_edif [options] [filename]
2
3  Write the current design to an EDIF netlist file.
4
5      -top top_module
6          set the specified module as design top module
7
8      -nogndvcc
9          do not create "GND" and "VCC" cells. (this will produce an error
10          if the design contains constant nets. use "hilomap" to map to custom
11          constant drivers first)
12
13 Unfortunately there are different "flavors" of the EDIF file format. This
14 command generates EDIF files for the Xilinx place&route tools. It might be
15 necessary to make small modifications to this command when a different tool
16 is targeted.
```

C.141   write_file – write a text to a file

```
1      write_file [options] output_file [input_file]
2
3  Write the text from the input file to the output file.
4
5      -a
6          Append to output file (instead of overwriting)
7
8
9  Inside a script the input file can also can a here-document:
10
11      write_file hello.txt <<EOT
12      Hello World!
13      EOT
```

C.142 write_ilang – write design to ilang file

```
 1      write_ilang [filename]
 2
 3  Write the current design to an 'ilang' file. (ilang is a text representation
 4  of a design in yosys's internal format.)
 5
 6      -selected
 7          only write selected parts of the design.
```

C.143 write_intersynth – write design to InterSynth netlist file

```
 1      write_intersynth [options] [filename]
 2
 3  Write the current design to an 'intersynth' netlist file. InterSynth is
 4  a tool for Coarse-Grain Example-Driven Interconnect Synthesis.
 5
 6      -notypes
 7          do not generate celltypes and conntypes commands. i.e. just output
 8          the netlists. this is used for postsilicon synthesis.
 9
10      -lib <verilog_or_ilang_file>
11          Use the specified library file for determining whether cell ports are
12          inputs or outputs. This option can be used multiple times to specify
13          more than one library.
14
15      -selected
16          only write selected modules. modules must be selected entirely or
17          not at all.
18
19  http://www.clifford.at/intersynth/
```

C.144 write_json – write design to a JSON file

```
 1      write_json [options] [filename]
 2
 3  Write a JSON netlist of the current design.
 4
 5      -aig
 6          include AIG models for the different gate types
 7
 8
 9  The general syntax of the JSON output created by this command is as follows:
10
11      {
12        "modules": {
13          <module_name>: {
14            "ports": {
15              <port_name>: <port_details>,
```

```
16            ...
17          },
18          "cells": {
19            <cell_name>: <cell_details>,
20            ...
21          },
22          "netnames": {
23            <net_name>: <net_details>,
24            ...
25          }
26        }
27      },
28      "models": {
29        ...
30      },
31    }

33  Where <port_details> is:

35      {
36        "direction": <"input" | "output" | "inout">,
37        "bits": <bit_vector>
38      }

40  And <cell_details> is:

42      {
43        "hide_name": <1 | 0>,
44        "type": <cell_type>,
45        "parameters": {
46          <parameter_name>: <parameter_value>,
47          ...
48        },
49        "attributes": {
50          <attribute_name>: <attribute_value>,
51          ...
52        },
53        "port_directions": {
54          <port_name>: <"input" | "output" | "inout">,
55          ...
56        },
57        "connections": {
58          <port_name>: <bit_vector>,
59          ...
60        },
61      }

63  And <net_details> is:

65      {
66        "hide_name": <1 | 0>,
67        "bits": <bit_vector>
68      }
69
```

```
70 | The "hide_name" fields are set to 1 when the name of this cell or net is
71 | automatically created and is likely not of interest for a regular user.
72 |
73 | The "port_directions" section is only included for cells for which the
74 | interface is known.
75 |
76 | Module and cell ports and nets can be single bit wide or vectors of multiple
77 | bits. Each individual signal bit is assigned a unique integer. The <bit_vector>
78 | values referenced above are vectors of this integers. Signal bits that are
79 | connected to a constant driver are denoted as string "0" or "1" instead of
80 | a number.
81 |
82 | For example the following Verilog code:
83 |
84 |     module test(input x, y);
85 |       (* keep *) foo #(.P(42), .Q(1337))
86 |           foo_inst (.A({x, y}), .B({y, x}), .C({4'd10, {4{x}}})));
87 |     endmodule
88 |
89 | Translates to the following JSON output:
90 |
91 |     {
92 |       "modules": {
93 |         "test": {
94 |           "ports": {
95 |             "x": {
96 |               "direction": "input",
97 |               "bits": [ 2 ]
98 |             },
99 |             "y": {
100 |               "direction": "input",
101 |               "bits": [ 3 ]
102 |             }
103 |           },
104 |           "cells": {
105 |             "foo_inst": {
106 |               "hide_name": 0,
107 |               "type": "foo",
108 |               "parameters": {
109 |                 "Q": 1337,
110 |                 "P": 42
111 |               },
112 |               "attributes": {
113 |                 "keep": 1,
114 |                 "src": "test.v:2"
115 |               },
116 |               "connections": {
117 |                 "C": [ 2, 2, 2, 2, "0", "1", "0", "1" ],
118 |                 "B": [ 2, 3 ],
119 |                 "A": [ 3, 2 ]
120 |               }
121 |             }
122 |           },
123 |           "netnames": {
```

```
124             "y": {
125                "hide_name": 0,
126                "bits": [ 3 ],
127                "attributes": {
128                   "src": "test.v:1"
129                }
130             },
131             "x": {
132                "hide_name": 0,
133                "bits": [ 2 ],
134                "attributes": {
135                   "src": "test.v:1"
136                }
137             }
138          }
139        }
140      }
141    }
142
143 The models are given as And-Inverter-Graphs (AIGs) in the following form:
144
145      "models": {
146        <model_name>: [
147          /*   0 */ [ <node-spec> ],
148          /*   1 */ [ <node-spec> ],
149          /*   2 */ [ <node-spec> ],
150          ...
151        ],
152        ...
153      },
154
155 The following node-types may be used:
156
157      [ "port", <portname>, <bitindex>, <out-list> ]
158         - the value of the specified input port bit
159
160      [ "nport", <portname>, <bitindex>, <out-list> ]
161         - the inverted value of the specified input port bit
162
163      [ "and", <node-index>, <node-index>, <out-list> ]
164         - the ANDed value of the specified nodes
165
166      [ "nand", <node-index>, <node-index>, <out-list> ]
167         - the inverted ANDed value of the specified nodes
168
169      [ "true", <out-list> ]
170         - the constant value 1
171
172      [ "false", <out-list> ]
173         - the constant value 0
174
175 All nodes appear in topological order. I.e. only nodes with smaller indices
176 are referenced by "and" and "nand" nodes.
177
```

```
178  The optional <out-list> at the end of a node specification is a list of
179  output portname and bitindex pairs, specifying the outputs driven by this node.
180
181  For example, the following is the model for a 3-input 3-output $reduce_and cell
182  inferred by the following code:
183
184      module test(input [2:0] in, output [2:0] out);
185        assign in = &out;
186      endmodule
187
188      "$reduce_and:3U:3": [
189        /*   0 */ [ "port", "A", 0 ],
190        /*   1 */ [ "port", "A", 1 ],
191        /*   2 */ [ "and", 0, 1 ],
192        /*   3 */ [ "port", "A", 2 ],
193        /*   4 */ [ "and", 2, 3, "Y", 0 ],
194        /*   5 */ [ "false", "Y", 1, "Y", 2 ]
195      ]
196
197  Future version of Yosys might add support for additional fields in the JSON
198  format. A program processing this format must ignore all unknown fields.
```

C.145   write_smt2 − write design to SMT-LIBv2 file

```
1       write_smt2 [options] [filename]
2
3   Write a SMT-LIBv2 [1] description of the current design. For a module with name
4   '<mod>' this will declare the sort '<mod>_s' (state of the module) and the
5   functions operating on that state.
6
7   The '<mod>_s' sort represents a module state. Additional '<mod>_n' functions
8   are provided that can be used to access the values of the signals in the module.
9   By default only ports, registers, and wires with the 'keep' attribute set are
10  made available via such functions. With the -nobv option, multi-bit wires are
11  exported as separate functions of type Bool for the individual bits. Without
12  -nobv multi-bit wires are exported as single functions of type BitVec.
13
14  The '<mod>_t' function evaluates to 'true' when the given pair of states
15  describes a valid state transition.
16
17  The '<mod>_a' function evaluates to 'true' when the given state satisfies
18  the asserts in the module.
19
20  The '<mod>_u' function evaluates to 'true' when the given state satisfies
21  the assumptions in the module.
22
23  The '<mod>_i' function evaluates to 'true' when the given state conforms
24  to the initial state. Furthermore the '<mod>_is' function should be asserted
25  to be true for initial states in addition to '<mod>_i', and should be
26  asserted to be false for non-initial states.
27
28  For hierarchical designs, the '<mod>_h' function must be asserted for each
```

```
 29  state to establish the design hierarchy. The '<mod>_h <cellname>' function
 30  evaluates to the state corresponding to the given cell within <mod>.
 31
 32      -verbose
 33          this will print the recursive walk used to export the modules.
 34
 35      -nobv
 36          disable support for BitVec (FixedSizeBitVectors theory). without this
 37          option multi-bit wires are represented using the BitVec sort and
 38          support for coarse grain cells (incl. arithmetic) is enabled.
 39
 40      -nomem
 41          disable support for memories (via ArraysEx theory). this option is
 42          implied by -nobv. only $mem cells without merged registers in
 43          read ports are supported. call "memory" with -nordff to make sure
 44          that no registers are merged into $mem read ports. '<mod>_m' functions
 45          will be generated for accessing the arrays that are used to represent
 46          memories.
 47
 48      -wires
 49          create '<mod>_n' functions for all public wires. by default only ports,
 50          registers, and wires with the 'keep' attribute are exported.
 51
 52      -tpl <template_file>
 53          use the given template file. the line containing only the token '%%'
 54          is replaced with the regular output of this command.
 55
 56  [1] For more information on SMT-LIBv2 visit http://smt-lib.org/ or read David
 57  R. Cok's tutorial: http://www.grammatech.com/resources/smt/SMTLIBTutorial.pdf
 58
 59  ------------------------------------------------------------------------
 60
 61  Example:
 62
 63  Consider the following module (test.v). We want to prove that the output can
 64  never transition from a non-zero value to a zero value.
 65
 66          module test(input clk, output reg [3:0] y);
 67              always @(posedge clk)
 68                  y <= (y << 1) | ^y;
 69          endmodule
 70
 71  For this proof we create the following template (test.tpl).
 72
 73          ; we need QF_UFBV for this poof
 74          (set-logic QF_UFBV)
 75
 76          ; insert the auto-generated code here
 77          %%
 78
 79          ; declare two state variables s1 and s2
 80          (declare-fun s1 () test_s)
 81          (declare-fun s2 () test_s)
 82
```

```
 83 |        ; state s2 is the successor of state s1
 84 |        (assert (test_t s1 s2))
 85 |
 86 |        ; we are looking for a model with y non-zero in s1
 87 |        (assert (distinct (|test_n y| s1) #b0000))
 88 |
 89 |        ; we are looking for a model with y zero in s2
 90 |        (assert (= (|test_n y| s2) #b0000))
 91 |
 92 |        ; is there such a model?
 93 |        (check-sat)
 94 |
 95 | The following yosys script will create a 'test.smt2' file for our proof:
 96 |
 97 |        read_verilog test.v
 98 |        hierarchy -check; proc; opt; check -assert
 99 |        write_smt2 -bv -tpl test.tpl test.smt2
100 |
101 | Running 'cvc4 test.smt2' will print 'unsat' because y can never transition
102 | from non-zero to zero in the test design.
```

C.146   write_smv – write design to SMV file

```
 1 |     write_smv [options] [filename]
 2 |
 3 | Write an SMV description of the current design.
 4 |
 5 |    -verbose
 6 |        this will print the recursive walk used to export the modules.
 7 |
 8 |    -tpl <template_file>
 9 |        use the given template file. the line containing only the token '%%'
10 |        is replaced with the regular output of this command.
11 |
12 | THIS COMMAND IS UNDER CONSTRUCTION
```

C.147   write_spice – write design to SPICE netlist file

```
 1 |     write_spice [options] [filename]
 2 |
 3 | Write the current design to an SPICE netlist file.
 4 |
 5 |    -big_endian
 6 |        generate multi-bit ports in MSB first order
 7 |        (default is LSB first)
 8 |
 9 |    -neg net_name
10 |        set the net name for constant 0 (default: Vss)
11 |
12 |    -pos net_name
```

```
13          set the net name for constant 1 (default: Vdd)
14
15      -nc_prefix
16          prefix for not-connected nets (default: _NC)
17
18      -inames
19          include names of internal ($-prefixed) nets in outputs
20          (default is to use net numbers instead)
21
22      -top top_module
23          set the specified module as design top module
```

C.148   write_verilog – write design to Verilog file

```
1       write_verilog [options] [filename]
2
3   Write the current design to a Verilog file.
4
5       -norename
6           without this option all internal object names (the ones with a dollar
7           instead of a backslash prefix) are changed to short names in the
8           format '_<number>_'.
9
10      -renameprefix <prefix>
11          insert this prefix in front of auto-generated instance names
12
13      -noattr
14          with this option no attributes are included in the output
15
16      -attr2comment
17          with this option attributes are included as comments in the output
18
19      -noexpr
20          without this option all internal cells are converted to Verilog
21          expressions.
22
23      -nodec
24          32-bit constant values are by default dumped as decimal numbers,
25          not bit pattern. This option decativates this feature and instead
26          will write out all constants in binary.
27
28      -nostr
29          Parameters and attributes that are specified as strings in the
30          original input will be output as strings by this back-end. This
31          decativates this feature and instead will write string constants
32          as binary numbers.
33
34      -defparam
35          Use 'defparam' statements instead of the Verilog-2001 syntax for
36          cell parameters.
37
38      -blackboxes
```

```
39          usually modules with the 'blackbox' attribute are ignored. with
40          this option set only the modules with the 'blackbox' attribute
41          are written to the output file.
42
43      -selected
44          only write selected modules. modules must be selected entirely or
45          not at all.
46
47      -v
48          verbose output (print new names of all renamed wires and cells)
49
50  Note that RTLIL processes can't always be mapped directly to Verilog
51  always blocks. This frontend should only be used to export an RTLIL
52  netlist, i.e. after the "proc" pass has been used to convert all
53  processes to logic networks and registers. A warning is generated when
54  this command is called on a design with RTLIL processes.
```

C.149   zinit – add inverters so all FF are zero-initialized

```
1      zinit [options] [selection]
2
3  Add inverters as needed to make all FFs zero-initialized.
4
5      -all
6          also add zero initialization to uninitialized FFs
```

Appendix D

Application Notes

This appendix contains copies of the Yosys application notes.

Yosys Application Note 010: Converting Verilog to BLIF

Clifford Wolf
November 2013

Abstract—Verilog-2005 is a powerful Hardware Description Language (HDL) that can be used to easily create complex designs from small HDL code. It is the preferred method of design entry for many designers[1].

The Berkeley Logic Interchange Format (BLIF) [6] is a simple file format for exchanging sequential logic between programs. It is easy to generate and easy to parse and is therefore the preferred method of design entry for many authors of logic synthesis tools.

Yosys [1] is a feature-rich Open-Source Verilog synthesis tool that can be used to bridge the gap between the two file formats. It implements most of Verilog-2005 and thus can be used to import modern behavioral Verilog designs into BLIF-based design flows without dependencies on proprietary synthesis tools.

The scope of Yosys goes of course far beyond Verilog logic synthesis. But it is a useful and important feature and this Application Note will focus on this aspect of Yosys.

I. Installation

Yosys written in C++ (using features from C++11) and is tested on modern Linux. It should compile fine on most UNIX systems with a C++11 compiler. The README file contains useful information on building Yosys and its prerequisites.

Yosys is a large and feature-rich program with a couple of dependencies. It is, however, possible to deactivate some of the dependencies in the Makefile, resulting in features in Yosys becoming unavailable. When problems with building Yosys are encountered, a user who is only interested in the features of Yosys that are discussed in this Application Note may deactivate TCL, `Qt` and `MiniSAT` support in the `Makefile` and may opt against building `yosys-abc`.

This Application Note is based on GIT Rev. `e216e0e` from 2013-11-23 of Yosys [1]. The Verilog sources used for the examples are taken from yosys-bigsim [2], a collection of real-world designs used for regression testing Yosys.

II. Getting Started

We start our tour with the Navré processor from yosys-bigsim. The Navré processor [3] is an Open Source AVR clone. It is a single module (`softusb_navre`) in a single design file (`softusb_navre.v`). It also is using only features that map nicely to the BLIF format, for example it only uses synchronous resets.

Converting `softusb_navre.v` to `softusb_navre.blif` could not be easier:

```
1  yosys -o softusb_navre.blif -S softusb_navre.v
```

Listing 1. Calling Yosys without script file

Behind the scenes Yosys is controlled by synthesis scripts that execute commands that operate on Yosys' internal state. For example, the `-o softusb_navre.blif` option just adds the command `write_blif softusb_navre.blif` to the end of the script. Likewise a file on the command line – `softusb_navre.v` –

---

[1]The other half prefers VHDL, a very different but – of course – equally powerful language.

in this case – adds the command `read_verilog softusb_navre.v` to the beginning of the synthesis script. In both cases the file type is detected from the file extension.

Finally the option `-S` instantiates a built-in default synthesis script. Instead of using `-S` one could also specify the synthesis commands for the script on the command line using the `-p` option, either using individual options for each command or by passing one big command string with a semicolon-separated list of commands. But in most cases it is more convenient to use an actual script file.

III. Using a Synthesis Script

With a script file we have better control over Yosys. The following script file replicates what the command from the last section did:

```
1  read_verilog softusb_navre.v
2  hierarchy
3  proc; opt; memory; opt; techmap; opt
4  write_blif softusb_navre.blif
```

Listing 2. `softusb_navre.ys`

The first and last line obviously read the Verilog file and write the BLIF file.

The 2nd line checks the design hierarchy and instantiates parametrized versions of the modules in the design, if necessary. In the case of this simple design this is a no-op. However, as a general rule a synthesis script should always contain this command as first command after reading the input files.

The 3rd line does most of the actual work:

- The command `opt` is the Yosys' built-in optimizer. It can perform some simple optimizations such as const-folding and removing unconnected parts of the design. It is common practice to call opt after each major step in the synthesis procedure. In cases where too much optimization is not appreciated (for example when analyzing a design), it is recommended to call `clean` instead of `opt`.
- The command `proc` converts *processes* (Yosys' internal representation of Verilog `always-` and `initial`-blocks) to circuits of multiplexers and storage elements (various types of flip-flops).
- The command `memory` converts Yosys' internal representations of arrays and array accesses to multi-port block memories, and then maps this block memories to address decoders and flip-flops, unless the option `-nomap` is used, in which case the multi-port block memories stay in the design and can then be mapped to architecture-specific memory primitives using other commands.
- The command `techmap` turns a high-level circuit with coarse grain cells such as wide adders and multipliers to a fine-grain circuit of simple logic primitives and single-bit storage elements. The command does that by substituting the complex cells by circuits of simpler cells. It is possible to provide a custom set of rules for this process in the form of a Verilog source file, as we will see in the next section.

Now Yosys can be run with the filename of the synthesis script as argument:

```
1  yosys softusb_navre.ys
```

Listing 3. Calling Yosys with script file

Now that we are using a synthesis script we can easily modify how Yosys synthesizes the design. The first thing we should customize is the call to the hierarchy command:

Whenever it is known that there are no implicit blackboxes in the design, i.e. modules that are referenced but are not defined, the hierarchy command should be called with the -check option. This will then cause synthesis to fail when implicit blackboxes are found in the design.

The 2nd thing we can improve regarding the hierarchy command is that we can tell it the name of the top level module of the design hierarchy. It will then automatically remove all modules that are not referenced from this top level module.

For many designs it is also desired to optimize the encodings for the finite state machines (FSMs) in the design. The fsm command finds FSMs, extracts them, performs some basic optimizations and then generate a circuit from the extracted and optimized description. It would also be possible to tell the fsm command to leave the FSMs in their extracted form, so they can be further processed using custom commands. But in this case we don't want that.

So now we have the final synthesis script for generating a BLIF file for the Navré CPU:

```
1  read_verilog softusb_navre.v
2  hierarchy -check -top softusb_navre
3  proc; opt; memory; opt; fsm; opt; techmap; opt
4  write_blif softusb_navre.blif
```

Listing 4. softusb_navre.ys (improved)

IV. Advanced Example: The Amber23 ARMv2a CPU

Our 2nd example is the Amber23 [4] ARMv2a CPU. Once again we base our example on the Verilog code that is included in yosys-bigsim [2].

The problem with this core is that it contains no dedicated reset logic. Instead the coding techniques shown in Listing 6 are used to define reset values for the global asynchronous reset in an FPGA

```
1  read_verilog a23_alu.v
2  read_verilog a23_barrel_shift_fpga.v
3  read_verilog a23_barrel_shift.v
4  read_verilog a23_cache.v
5  read_verilog a23_coprocessor.v
6  read_verilog a23_core.v
7  read_verilog a23_decode.v
8  read_verilog a23_execute.v
9  read_verilog a23_fetch.v
10 read_verilog a23_multiply.v
11 read_verilog a23_ram_register_bank.v
12 read_verilog a23_register_bank.v
13 read_verilog a23_wishbone.v
14 read_verilog generic_sram_byte_en.v
15 read_verilog generic_sram_line_en.v
16 hierarchy -check -top a23_core
17 add -global_input globrst 1
18 proc -global_arst globrst
19 techmap -map adff2dff.v
20 opt; memory; opt; fsm; opt; techmap
21 write_blif amber23.blif
```

Listing 5. amber23.ys

```
1  reg [7:0] a = 13, b;
2  initial b = 37;
```

Listing 6. Implicit coding of global asynchronous resets

implementation. This design can not be expressed in BLIF as it is. Instead we need to use a synthesis script that transforms this form to synchronous resets that can be expressed in BLIF.

(Note that there is no problem if this coding techniques are used to model ROM, where the register is initialized using this syntax but is never updated otherwise.)

Listing 5 shows the synthesis script for the Amber23 core. In line 17 the add command is used to add a 1-bit wide global input signal with the name globrst. That means that an input with that name is added to each module in the design hierarchy and then all module instantiations are altered so that this new signal is connected throughout the whole design hierarchy.

In line 18 the proc command is called. But in this script the signal name globrst is passed to the command as a global reset signal for resetting the registers to their assigned initial values.

Finally in line 19 the techmap command is used to replace all instances of flip-flops with asynchronous resets with flip-flops with synchronous resets. The map file used for this is shown in Listing 7. Note how the techmap_celltype attribute is used in line 1 to tell the techmap command which cells to replace in the design, how the _TECHMAP_FAIL_ wire in lines 15 and 16 (which evaluates to a constant value) determines if the parameter set is compatible with this replacement circuit, and how the _TECHMAP_DO_ wire in line 13 provides a mini synthesis-script to be used to process this cell.

```
1  (* techmap_celltype = "$adff" *)
2  module adff2dff (CLK, ARST, D, Q);
3
4  parameter WIDTH = 1;
5  parameter CLK_POLARITY = 1;
6  parameter ARST_POLARITY = 1;
7  parameter ARST_VALUE = 0;
8
9  input CLK, ARST;
10 input [WIDTH-1:0] D;
11 output reg [WIDTH-1:0] Q;
12
13 wire [1023:0] _TECHMAP_DO_ = "proc";
14
15 wire _TECHMAP_FAIL_ =
16     !CLK_POLARITY || !ARST_POLARITY;
17
18 always @(posedge CLK)
19        if (ARST)
20                Q <= ARST_VALUE;
21        else
22                Q <= D;
23
24 endmodule
```

Listing 7. adff2dff.v

```c
#include <stdint.h>
#include <stdbool.h>

#define BITMAP_SIZE 64
#define OUTPORT 0x10000000

static uint32_t bitmap[BITMAP_SIZE/32];

static void bitmap_set(uint32_t idx) { bitmap[idx/32] |= 1 << (idx % 32); }
static bool bitmap_get(uint32_t idx) { return (bitmap[idx/32] & (1 << (idx % 32))) != 0; }
static void output(uint32_t val) { *((volatile uint32_t*)OUTPORT) = val; }

int main() {
        uint32_t i, j, k;
        output(2);
        for (i = 0; i < BITMAP_SIZE; i++) {
                if (bitmap_get(i)) continue;
                output(3+2*i);
                for (j = 2*(3+2*i);; j += 3+2*i) {
                        if (j%2 == 0) continue;
                        k = (j-3)/2;
                        if (k >= BITMAP_SIZE) break;
                        bitmap_set(k);
                }
        }
        output(0);
        return 0;
}
```

Listing 8. Test program for the Amber23 CPU (Sieve of Eratosthenes). Compiled using GCC 4.6.3 for ARM with `-Os -marm -march=armv2a -mno-thumb-interwork -ffreestanding`, linked with `--fix-v4bx` set and booted with a custom setup routine written in ARM assembler.

V. VERIFICATION OF THE AMBER23 CPU

The BLIF file for the Amber23 core, generated using Listings 5 and 7 and the version of the Amber23 RTL source that is bundled with yosys-bigsim, was verified using the test-bench from yosys-bigsim. It successfully executed the program shown in Listing 8 in the test-bench.

For simulation the BLIF file was converted back to Verilog using ABC [5]. So this test includes the successful transformation of the BLIF file into ABC's internal format as well.

The only thing left to write about the simulation itself is that it probably was one of the most energy inefficient and time consuming ways of successfully calculating the first 31 primes the author has ever conducted.

VI. LIMITATIONS

At the time of this writing Yosys does not support multi-dimensional memories, does not support writing to individual bits of array elements, does not support initialization of arrays with `$readmemb` and `$readmemh`, and has only limited support for tristate logic, to name just a few limitations.

That being said, Yosys can synthesize an overwhelming majority of real-world Verilog RTL code. The remaining cases can usually be modified to be compatible with Yosys quite easily.

The various designs in yosys-bigsim are a good place to look for examples of what is within the capabilities of Yosys.

VII. CONCLUSION

Yosys is a feature-rich Verilog-2005 synthesis tool. It has many uses, but one is to provide an easy gateway from high-level Verilog code to low-level logic circuits.

The command line option `-S` can be used to quickly synthesize Verilog code to BLIF files without a hassle.

With custom synthesis scripts it becomes possible to easily perform high-level optimizations, such as re-encoding FSMs. In some extreme cases, such as the Amber23 ARMv2 CPU, the more advanced Yosys features can be used to change a design to fit a certain need without actually touching the RTL code.

REFERENCES

[1] Clifford Wolf. The Yosys Open SYnthesis Suite.
http://www.clifford.at/yosys/
[2] yosys-bigsim, a collection of real-world Verilog designs for regression testing purposes.
https://github.com/cliffordwolf/yosys-bigsim
[3] Sebastien Bourdeauducq. Navré AVR clone (8-bit RISC).
http://opencores.org/project,navre
[4] Conor Santifort. Amber ARM-compatible core.
http://opencores.org/project,amber
[5] Berkeley Logic Synthesis and Verification Group. ABC: A System for Sequential Synthesis and Verification.
http://www.eecs.berkeley.edu/~alanmi/abc/
[6] Berkeley Logic Interchange Format (BLIF)
http://vlsi.colorado.edu/~vis/blif.ps

Yosys Application Note 011: Interactive Design Investigation

Clifford Wolf
Original Version December 2013

Abstract—Yosys [1] can be a great environment for building custom synthesis flows. It can also be an excellent tool for teaching and learning Verilog based RTL synthesis. In both applications it is of great importance to be able to analyze the designs it produces easily.

This Yosys application note covers the generation of circuit diagrams with the Yosys show **command, the selection of interesting parts of the circuit using the** select **command, and briefly discusses advanced investigation commands for evaluating circuits and solving SAT problems.**

I. Installation and Prerequisites

This Application Note is based on the Yosys [1] GIT Rev. 2b90ba1 from 2013-12-08. The README file covers how to install Yosys. The show command requires a working installation of GraphViz [2] and [3] for generating the actual circuit diagrams.

II. Overview

This application note is structured as follows:

Sec. III introduces the show command and explains the symbols used in the circuit diagrams generated by it.

Sec. IV introduces additional commands used to navigate in the design, select portions of the design, and print additional information on the elements in the design that are not contained in the circuit diagrams.

Sec. V introduces commands to evaluate the design and solve SAT problems within the design.

Sec. VI concludes the document and summarizes the key points.

III. Introduction to the show command

The show command generates a circuit diagram for the design in its current state. Various options can be used to change the appearance of the circuit diagram, set the name and format for the output file, and so forth. When called without any special options, it saves the circuit diagram in a temporary file and launches xdot to display the diagram. Subsequent calls to show re-use the xdot instance (if still running).

```
1   $ cat example.ys
2   read_verilog example.v
3   show -pause
4   proc
5   show -pause
6   opt
7   show -pause
8
9   $ cat example.v
10  module example(input clk, a, b, c,
11                  output reg [1:0] y);
12      always @(posedge clk)
13          if (c)
14              y <= c ? a + b : 2'd0;
15  endmodule
```

Figure 1. Yosys script with show commands and example design

A. A simple circuit

Fig. 1 shows a simple synthesis script and a Verilog file that demonstrate the usage of show in a simple setting. Note that show is called with the -pause option, that halts execution of the Yosys script until the user presses the Enter key. The show -pause command also allows the user to enter an interactive shell to further investigate the circuit before continuing synthesis.

So this script, when executed, will show the design after each of the three synthesis commands. The generated circuit diagrams are shown in Fig. 2.

The first diagram (from top to bottom) shows the design directly after being read by the Verilog front-end. Input and output ports are displayed as octagonal shapes. Cells are displayed as rectangles with inputs on the left and outputs on the right side. The cell labels are two lines long: The first line contains a unique identifier for the cell and the second line contains the cell type. Internal cell types are prefixed with a dollar sign. The Yosys manual contains a chapter on the internal cell library used in Yosys.

Constants are shown as ellipses with the constant value as label. The syntax <bit_width>'<bits> is used for for constants that are not 32-bit wide and/or contain bits that are not 0 or 1 (i.e. x or z). Ordinary 32-bit constants are written using decimal numbers.

Single-bit signals are shown as thin arrows pointing from the driver to the load. Signals that are multiple bits wide are shown as think arrows.

Finally *processes* are shown in boxes with round corners. Processes are Yosys' internal representation of the decision-trees and synchronization events modelled in a Verilog always-block. The label reads PROC followed by a unique identifier in the first line and contains the source code location of the original always-block in the 2nd line. Note how the multiplexer from the ?:-expression
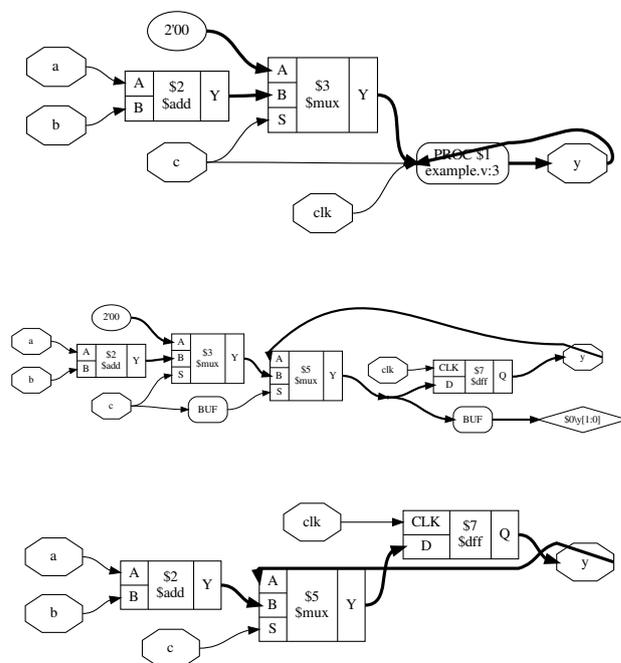

Figure 2. Output of the three show commands from Fig. 1

is represented as a $mux cell but the multiplexer from the if-statement is yet still hidden within the process.

The proc command transforms the process from the first diagram into a multiplexer and a d-type flip-flip, which brings us to the 2nd diagram.

The Rhombus shape to the right is a dangling wire. (Wire nodes are only shown if they are dangling or have "public" names, for example names assigned from the Verilog input.) Also note that the design now contains two instances of a BUF-node. This are artefacts left behind by the proc-command. It is quite usual to see such artefacts after calling commands that perform changes in the design, as most commands only care about doing the transformation in the least complicated way, not about cleaning up after them. The next call to clean (or opt, which includes clean as one of its operations) will clean up this artefacts. This operation is so common in Yosys scripts that it can simply be abbreviated with the ;; token, which doubles as separator for commands. Unless one wants to specifically analyze this artefacts left behind some operations, it is therefore recommended to always call clean before calling show.

In this script we directly call opt as next step, which finally leads us to the 3rd diagram in Fig. 2. Here we see that the opt command not only has removed the artifacts left behind by proc, but also determined correctly that it can remove the first $mux cell without changing the behavior of the circuit.

B. Break-out boxes for signal vectors

As has been indicated by the last example, Yosys is can manage signal vectors (aka. multi-bit wires or buses) as native objects. This provides great advantages when analyzing circuits that operate on wide integers. But it also introduces some additional complexity
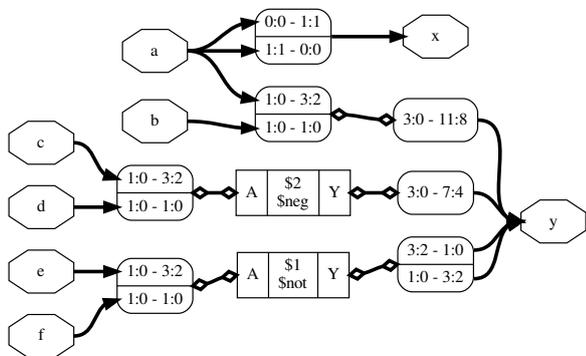
Figure 3. Output of yosys -p 'proc; opt; show' splice.v

```
1  module splice_demo(a, b, c, d, e, f, x, y);
2
3  input [1:0] a, b, c, d, e, f;
4  output [1:0] x = {a[0], a[1]};
5
6  output [11:0] y;
7  assign {y[11:4], y[1:0], y[3:2]} =
8                   {a, b, -{c, d}, ~{e, f}};
9
10 endmodule
```

Figure 4. splice.v
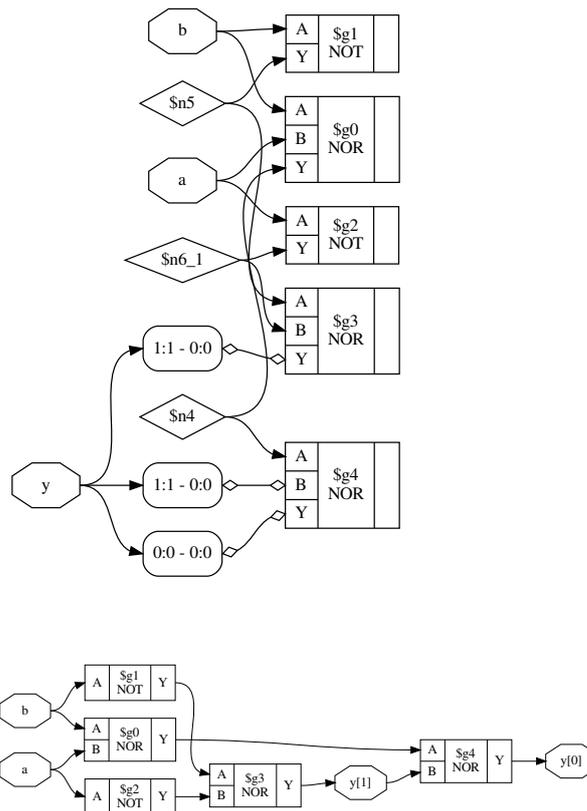
Figure 5. Effects of splitnets command and of providing a cell library. (The circuit is a half-adder built from simple CMOS gates.)

when the individual bits of of a signal vector are accessed. The example show in Fig. 3 and 4 demonstrates how such circuits are visualized by the show command.

The key elements in understanding this circuit diagram are of course the boxes with round corners and rows labeled <MSB_LEFT>:<LSB_LEFT> – <MSB_RIGHT>:<LSB_RIGHT>. Each of this boxes has one signal per row on one side and a common signal for all rows on the other side. The <MSB>:<LSB> tuples specify which bits of the signals are broken out and connected. So the top row of the box connecting the signals a and x indicates that the bit 0 (i.e. the range 0:0) from signal a is connected to bit 1 (i.e. the range 1:1) of signal x.

Lines connecting such boxes together and lines connecting such boxes to cell ports have a slightly different look to emphasise that they are not actual signal wires but a necessity of the graphical representation. This distinction seems like a technicality, until one wants to debug a problem related to the way Yosys internally represents signal vectors, for example when writing custom Yosys commands.

C. Gate level netlists

Finally Fig. 5 shows two common pitfalls when working with designs mapped to a cell library. The top figure has two problems: First Yosys did not have access to the cell library when this diagram was generated, resulting in all cell ports defaulting to being inputs. This is why all ports are drawn on the left side the cells are

awkwardly arranged in a large column. Secondly the two-bit vector y requires breakout-boxes for its individual bits, resulting in an unnecessary complex diagram.

For the 2nd diagram Yosys has been given a description of the cell library as Verilog file containing blackbox modules. There are two ways to load cell descriptions into Yosys: First the Verilog file for the cell library can be passed directly to the `show` command using the `-lib <filename>` option. Secondly it is possible to load cell libraries into the design with the `read_verilog -lib <filename>` command. The 2nd method has the great advantage that the library only needs to be loaded once and can then be used in all subsequent calls to the `show` command.

In addition to that, the 2nd diagram was generated after `split-net -ports` was run on the design. This command splits all signal vectors into individual signal bits, which is often desirable when looking at gate-level circuits. The `-ports` option is required to also split module ports. Per default the command only operates on interior signals.

D. Miscellaneous notes

Per default the `show` command outputs a temporary `dot` file and launches `xdot` to display it. The options `-format`, `-viewer` and `-prefix` can be used to change format, viewer and filename prefix. Note that the `pdf` and `ps` format are the only formats that support plotting multiple modules in one run.

In densely connected circuits it is sometimes hard to keep track of the individual signal wires. For this cases it can be useful to call `show` with the `-colors <integer>` argument, which randomly assigns colors to the nets. The integer (> 0) is used as seed value for the random color assignments. Sometimes it is necessary it try some values to find an assignment of colors that looks good.

The command `help show` prints a complete listing of all options supported by the `show` command.

IV. NAVIGATING THE DESIGN

Plotting circuit diagrams for entire modules in the design brings us only helps in simple cases. For complex modules the generated circuit diagrams are just stupidly big and are no help at all. In such cases one first has to select the relevant portions of the circuit.

In addition to *what* to display one also needs to carefully decide *when* to display it, with respect to the synthesis flow. In general it is a good idea to troubleshoot a circuit in the earliest state in which a problem can be reproduced. So if, for example, the internal state before calling the `techmap` command already fails to verify, it is better to troubleshoot the coarse-grain version of the circuit before `techmap` than the gate-level circuit after `techmap`.

Note: It is generally recommended to verify the internal state of a design by writing it to a Verilog file using `write_verilog -noexpr` and using the simulation models from `simlib.v` and `simcells.v` from the Yosys data directory (as printed by `yosys-config --datdir`).

A. Interactive Navigation

Once the right state within the synthesis flow for debugging the circuit has been identified, it is recommended to simply add the `shell` command to the matching place in the synthesis script. This command will stop the synthesis at the specified moment and go to shell mode, where the user can interactively enter commands.

For most cases, the shell will start with the whole design selected (i.e. when the synthesis script does not already narrow the selection). The command `ls` can now be used to create a list

```
1  yosys> ls
2
3  1 modules:
4    example
5
6  yosys> cd example
7
8  yosys [example]> ls
9
10  7 wires:
11    $0\y[1:0]
12    $add$example.v:5$2_Y
13    a
14    b
15    c
16    clk
17    y
18
19  3 cells:
20    $add$example.v:5$2
21    $procdff$7
22    $procmux$5
```

Figure 6. Demonstration of `ls` and `cd` using `example.v` from Fig. 1

of all modules. The command `cd` can be used to switch to one of the modules (type `cd ..` to switch back). Now the `ls` command lists the objects within that module. Fig. 6 demonstrates this using the design from Fig. 1.

There is a thing to note in Fig. 6: We can see that the cell names from Fig. 2 are just abbreviations of the actual cell names, namely the part after the last dollar-sign. Most auto-generated names (the ones starting with a dollar sign) are rather long and contains some additional information on the origin of the named object. But in most cases those names can simply be abbreviated using the last part.

Usually all interactive work is done with one module selected using the `cd` command. But it is also possible to work from the design-context (`cd ..`). In this case all object names must be prefixed with `<module_name>/`. For example `a*/b*` would refer to all objects whose names start with b from all modules whose names start with a.

The `dump` command can be used to print all information about an object. For example `dump $2` will print Fig. 7. This can for example be useful to determine the names of nets connected to cells, as the net-names are usually suppressed in the circuit diagram if

```
1   attribute \src "example.v:5"
2   cell $add $add$example.v:5$2
3     parameter \A_SIGNED 0
4     parameter \A_WIDTH 1
5     parameter \B_SIGNED 0
6     parameter \B_WIDTH 1
7     parameter \Y_WIDTH 2
8     connect \A \a
9     connect \B \b
10    connect \Y $add$example.v:5$2_Y
11  end
```

Figure 7. Output of `dump $2` using the design from Fig. 1 and Fig. 2
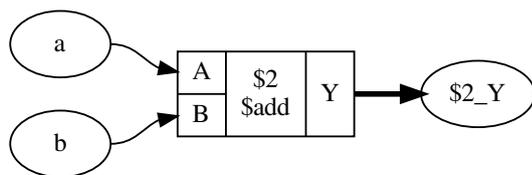
Figure 8. Output of show after select $2 or select t:$add (see also Fig. 2)

```
1   module sumprod(a, b, c, sum, prod);
2
3     input [7:0] a, b, c;
4     output [7:0] sum, prod;
5
6     {* sumstuff *}
7     assign sum = a + b + c;
8     {* *}
9
10    assign prod = a * b * c;
11
12  endmodule
```

Figure 10. Another test module for operations on selections

they are auto-generated.

For the remainder of this document we will assume that the commands are run from module-context and not design-context.

B. Working with selections

When a module is selected using the cd command, all commands (with a few exceptions, such as the read_* and write_* commands) operate only on the selected module. This can also be useful for synthesis scripts where different synthesis strategies should be applied to different modules in the design.

But for most interactive work we want to further narrow the set of selected objects. This can be done using the select command.

For example, if the command select $2 is executed, a subsequent show command will yield the diagram shown in Fig. 8. Note that the nets are now displayed in ellipses. This indicates that they are not selected, but only shown because the diagram contains a cell that is connected to the net. This of course makes no difference for the circuit that is shown, but it can be a useful information when manipulating selections.

Objects can not only be selected by their name but also by other properties. For example select t:$add will select all cells of type $add. In this case this is also yields the diagram shown in Fig. 8.

The output of help select contains a complete syntax reference for matching different properties.

Many commands can operate on explicit selections. For example the command dump t:$add will print information on all $add cells in the active module. Whenever a command has [selection] as last argument in its usage help, this means that it will use the engine behind the select command to evaluate additional arguments and use the resulting selection instead of the selection created by the last select command.

Normally the select command overwrites a previous selection. The commands select -add and select -del can be used to add or remove objects from the current selection.

The command select -clear can be used to reset the selection to the default, which is a complete selection of everything in the current module.

C. Operations on selections

The select command is actually much more powerful than it might seem on the first glimpse. When it is called with multiple arguments, each argument is evaluated and pushed separately on a stack. After all arguments have been processed it simply creates the union of all elements on the stack. So the following command will select all $add cells and all objects with the foo attribute set:

```
select t:$add a:foo
```

(Try this with the design shown in Fig. 9. Use the select -list command to list the current selection.)

In many cases simply adding more and more stuff to the selection is an ineffective way of selecting the interesting part of the design. Special arguments can be used to combine the elements on the stack. For example the %i arguments pops the last two elements from the stack, intersects them, and pushes the result back on the stack. So the following command will select all $add cells that have the foo attribute set:

```
1   module foobaraddsub(a, b, c, d, fa, fs, ba, bs);
2     input [7:0] a, b, c, d;
3     output [7:0] fa, fs, ba, bs;
4     assign fa = a + (* foo *) b;
5     assign fs = a - (* foo *) b;
6     assign ba = c + (* bar *) d;
7     assign bs = c - (* bar *) d;
8   endmodule
```

Figure 9. Test module for operations on selections
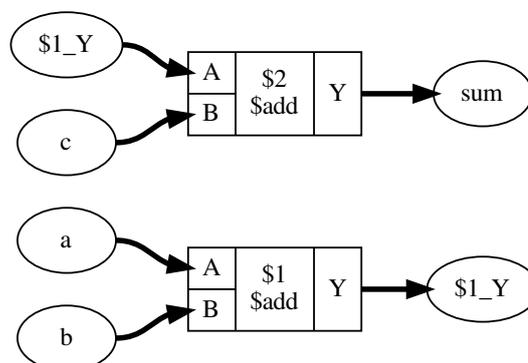

Figure 11. Output of show a:sumstuff on Fig. 10
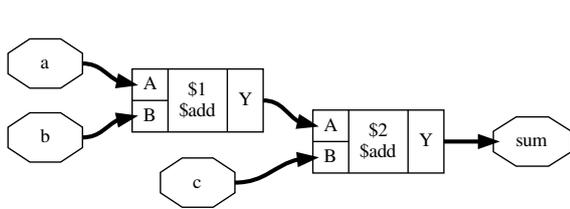
Figure 12. Output of `show a:sumstuff %x` on Fig. 10

```
select t:$add a:foo %i
```

The listing in Fig. 10 uses the Yosys non-standard `{* ... *}` syntax to set the attribute `sumstuff` on all cells generated by the first assign statement. (This works on arbitrary large blocks of Verilog code an can be used to mark portions of code for analysis.)

Selecting `a:sumstuff` in this module will yield the circuit diagram shown in Fig. 11. As only the cells themselves are selected, but not the temporary wire `$1_Y`, the two adders are shown as two disjunct parts. This can be very useful for global signals like clock and reset signals: just unselect them using a command such as `select -del clk rst` and each cell using them will get its own net label.

In this case however we would like to see the cells connected properly. This can be achieved using the `%x` action, that broadens the selection, i.e. for each selected wire it selects all cells connected to the wire and vice versa. So `show a:sumstuff %x` yields the diagram shown in Fig. 12.

D. Selecting logic cones

Fig. 12 shows what is called the *input cone* of `sum`, i.e. all cells and signals that are used to generate the signal `sum`. The `%ci` action can be used to select the input cones of all object in the top selection in the stack maintained by the `select` command.

As the `%x` action, this commands broadens the selection by one "step". But this time the operation only works against the direction of data flow. That means, wires only select cells via output ports and cells only select wires via input ports.

Fig. 13 show the sequence of diagrams generated by the following commands:

```
show prod
show prod %ci
show prod %ci %ci
show prod %ci %ci %ci
```

When selecting many levels of logic, repeating `%ci` over and over again can be a bit dull. So there is a shortcut for that: the number of iterations can be appended to the action. So for example the action `%ci3` is identical to performing the `%ci` action three times.

The action `%ci*` performs the `%ci` action over and over again until it has no effect anymore.

In most cases there are certain cell types and/or ports that should not be considered for the `%ci` action, or we only want to follow certain cell types and/or ports. This can be achieved using additional patterns that can be appended to the `%ci` action.

Lets consider the design from Fig. 14. It serves no purpose other than being a non-trivial circuit for demonstrating some of the
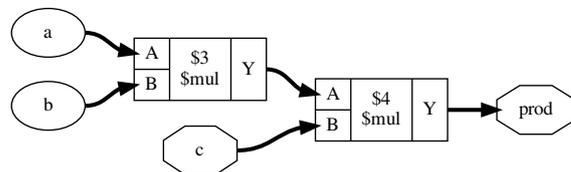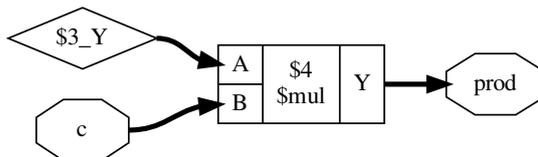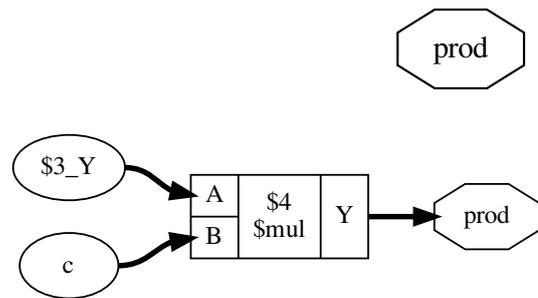
Figure 13. Objects selected by `select prod %ci...`

advanced Yosys features. We synthesize the circuit using `proc`; `opt`; `memory`; `opt` and change to the `memdemo` module with `cd memdemo`. If we type `show` now we see the diagram shown in Fig. 15.

But maybe we are only interested in the tree of multiplexers that select the output value. In order to get there, we would start by just showing the output signal and its immediate predecessors:

```
1  module memdemo(clk, d, y);
2
3  input clk;
4  input [3:0] d;
5  output reg [3:0] y;
6
7  integer i;
8  reg [1:0] s1, s2;
9  reg [3:0] mem [0:3];
10
11 always @(posedge clk) begin
12     for (i = 0; i < 4; i = i+1)
13         mem[i] <= mem[(i+1) % 4] + mem[(i+2) % 4];
14     { s2, s1 } = d ? { s1, s2 } ^ d : 4'b0;
15     mem[s1] <= d;
16     y <= mem[s2];
17 end
18
19 endmodule
```

Figure 14. Demo circuit for demonstrating some advanced Yosys features
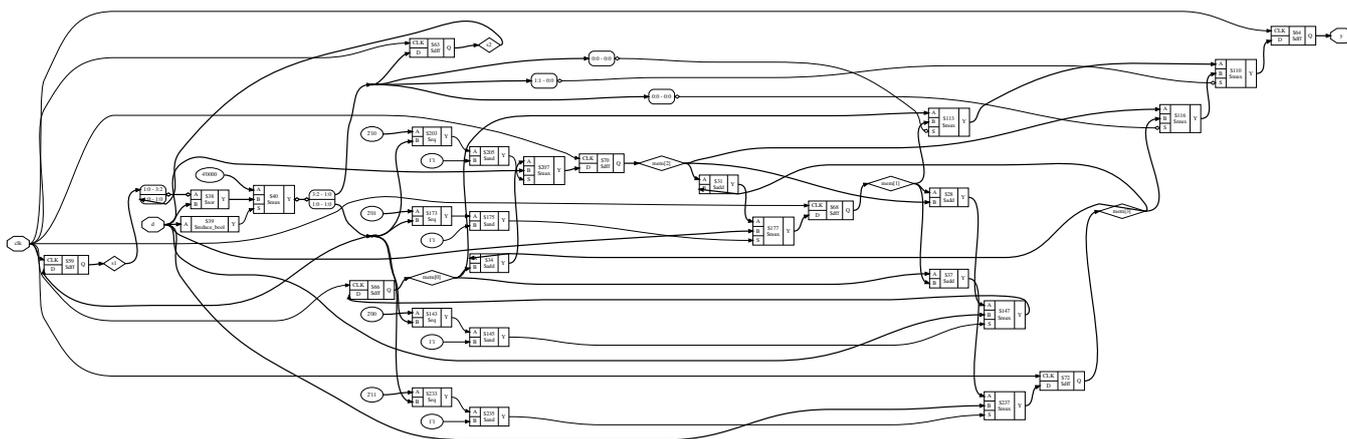
Figure 15. Complete circuit diagram for the design shown in Fig. 14

```
show y %ci2
```

From this we would learn that y is driven by a $dff cell, that y is connected to the output port Q, that the clk signal goes into the CLK input port of the cell, and that the data comes from a auto-generated wire into the input D of the flip-flop cell.

As we are not interested in the clock signal we add an additional pattern to the %ci action, that tells it to only follow ports Q and D of $dff cells:

```
show y %ci2:+$dff[Q,D]
```

To add a pattern we add a colon followed by the pattern to the %ci action. The pattern it self starts with – or +, indicating if it is an include or exclude pattern, followed by an optional comma separated list of cell types, followed by an optional comma separated list of port names in square brackets.

Since we know that the only cell considered in this case is a $dff cell, we could as well only specify the port names:

```
show y %ci2:+[Q,D]
```

Or we could decide to tell the %ci action to not follow the CLK input:

```
show y %ci2:-[CLK]
```

Next we would investigate the next logic level by adding another %ci2 to the command:
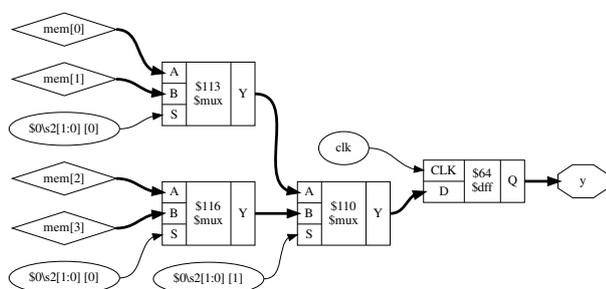
Figure 16. Output of show y %ci2:+$dff[Q,D] %ci*:-$mux[S]:-$dff

```
show y %ci2:-[CLK] %ci2
```

From this we would learn that the next cell is a $mux cell and we would add additional pattern to narrow the selection on the path we are interested. In the end we would end up with a command such as

```
show y %ci2:+$dff[Q,D] %ci*:-$mux[S]:-$dff
```

in which the first %ci jumps over the initial d-type flip-flop and the 2nd action selects the entire input cone without going over multiplexer select inputs and flip-flop cells. The diagram produces by this command is shown in Fig. 16.

Similar to %ci exists an action %co to select output cones that accepts the same syntax for pattern and repetition. The %x action mentioned previously also accepts this advanced syntax.

This actions for traversing the circuit graph, combined with the actions for boolean operations such as intersection (%i) and difference (%d) are powerful tools for extracting the relevant portions of the circuit under investigation.

See help select for a complete list of actions available in selections.

E. Storing and recalling selections

The current selection can be stored in memory with the command select -set <name>. It can later be recalled using select @<name>. In fact, the @<name> expression pushes the stored selection on the stack maintained by the select command. So for example

```
select @foo @bar %i
```

will select the intersection between the stored selections foo and bar.

In larger investigation efforts it is highly recommended to maintain a script that sets up relevant selections, so they can easily be recalled, for example when Yosys needs to be re-run after a design or source code change.

The history command can be used to list all recent interactive commands. This feature can be useful for creating such a script from the commands used in an interactive session.

V. Advanced investigation techniques

When working with very large modules, it is often not enough to just select the interesting part of the module. Instead it can be useful to extract the interesting part of the circuit into a separate module. This can for example be useful if one wants to run a series of synthesis commands on the critical part of the module and wants to carefully read all the debug output created by the commands in order to spot a problem. This kind of troubleshooting is much easier if the circuit under investigation is encapsulated in a separate module.

Fig. 17 shows how the `submod` command can be used to split the circuit from Fig. 14 and 15 into its components. The `-name` option is used to specify the name of the new module and also the name of the new cell in the current module.

A. Evaluation of combinatorial circuits

The `eval` command can be used to evaluate combinatorial circuits. For example (see Fig. 17 for the circuit diagram of `selstage`):

```
yosys [selstage]> eval -set s2,s1 4'b1001 -set d 4'hc -show n2

9. Executing EVAL pass (evaluate the circuit given an input).
Full command line: eval -set s2,s1 4'b1001 -set d 4'hc -show n2
 Eval result: \n2 = 2'10.
 Eval result: \n1 = 2'10.
```

So the `-set` option is used to set input values and the `-show` option is used to specify the nets to evaluate. If no `-show` option is specified, all selected output ports are used per default.

If a necessary input value is not given, an error is produced. The option `-set-undef` can be used to instead set all unspecified input nets to undef (x).

The `-table` option can be used to create a truth table. For example:

```
yosys [selstage]> eval -set-undef -set d[3:1] 0 -table s1,d[0]

10. Executing EVAL pass (evaluate the circuit given an input).
Full command line: eval -set-undef -set d[3:1] 0 -table s1,d[0]

  \s1 \d [0] |  \n1   \n2
  ---- ------ | ---- ----
  2'00    1'0 | 2'00 2'00
  2'00    1'1 | 2'xx 2'00
  2'01    1'0 | 2'00 2'00
  2'01    1'1 | 2'xx 2'01
  2'10    1'0 | 2'00 2'00
  2'10    1'1 | 2'xx 2'10
  2'11    1'0 | 2'00 2'00
  2'11    1'1 | 2'xx 2'11

 Assumed undef (x) value for the following signals: \s2
```

Note that the `eval` command (as well as the `sat` command discussed in the next sections) does only operate on flattened modules. It can not analyze signals that are passed through design hierarchy levels. So the `flatten` command must be used on modules that instantiate other modules before this commands can be applied.

B. Solving combinatorial SAT problems

Often the opposite of the `eval` command is needed, i.e. the circuits output is given and we want to find the matching input signals. For small circuits with only a few input bits this can be accomplished by trying all possible input combinations, as it is done by the `eval -table` command. For larger circuits however, Yosys provides the `sat` command that uses a SAT [4] solver [5] to solve this kind of problems.
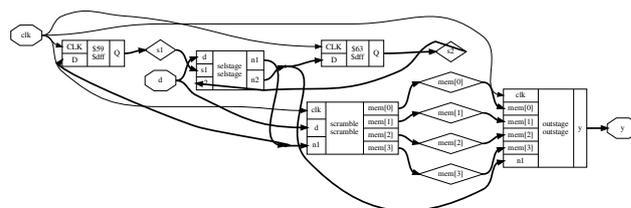
memdemo
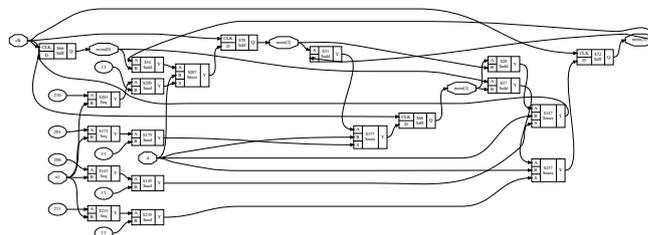
scramble
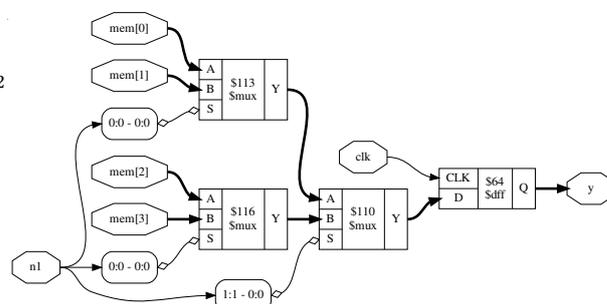
outstage
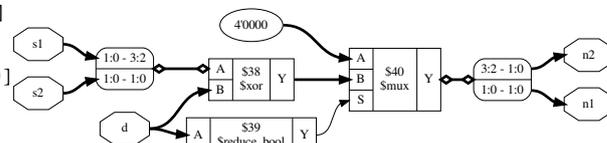
selstage

```
1 || select -set outstage y %ci2:+$dff[Q,D] %ci*:-$mux[S]:-$dff
2 || select -set selstage y %ci2:+$dff[Q,D] %ci*:-$dff @outstage %d
3 || select -set scramble mem* %ci2 %ci*:-$dff mem* %d @selstage %d
4 || submod -name scramble @scramble
5 || submod -name outstage @outstage
6 || submod -name selstage @selstage
```

Figure 17. The circuit from Fig. 14 and 15 broken up using `submod`

The `sat` command works very similar to the `eval` command. The main difference is that it is now also possible to set output values and find the corresponding input values. For Example:

```
yosys [selstage]> sat -show s1,s2,d -set s1 s2 -set n2,n1 4'b1001

11. Executing SAT pass (solving SAT problems in the circuit).
Full command line: sat -show s1,s2,d -set s1 s2 -set n2,n1 4'b1001

 Setting up SAT problem:
 Import set-constraint: \s1 = \s2
 Import set-constraint: { \n2 \n1 } = 4'1001
Final constraint equation: { \n2 \n1 \s1 } = { 4'1001 \s2 }
 Imported 3 cells to SAT database.
 Import show expression: { \s1 \s2 \d }
```

```
Solving problem with 81 variables and 207 clauses..
SAT solving finished - model found:

Signal Name                Dec        Hex           Bin
-------------------------- ---------- ---------- -------------
\d                            9          9           1001
\s1                           0          0             00
\s2                           0          0             00
```

Note that the `sat` command supports signal names in both arguments to the `-set` option. In the above example we used `-set s1 s2` to constraint `s1` and `s2` to be equal. When more complex constraints are needed, a wrapper circuit must be constructed that checks the constraints and signals if the constraint was met using an extra output port, which then can be forced to a value using the `-set` option. (Such a circuit that contains the circuit under test plus additional constraint checking circuitry is called a *miter* circuit.)

Fig. 18 shows a miter circuit that is supposed to be used as a prime number test. If `ok` is 1 for all input values `a` and `b` for a given `p`, then `p` is prime, or at least that is the idea.

The Yosys shell session shown in Fig. 19 demonstrates that SAT solvers can even find the unexpected solutions to a problem: Using integer overflow there actually is a way of "factorizing" 31. The clean solution would of course be to perform the test in 32 bits, for example by replacing `p != a*b` in the miter with `p != {16'd0,a}*b`, or by using a temporary variable for the 32 bit product `a*b`. But as 31 fits well into 8 bits (and as the purpose of this document is to show off Yosys features) we can also simply force the upper 8 bits of `a` and `b` to zero for the `sat` call, as is done in the second command in Fig. 19 (line 31).

The `-prove` option used in this example works similar to `-set`, but tries to find a case in which the two arguments are not equal. If such a case is not found, the property is proven to hold for all inputs that satisfy the other constraints.

It might be worth noting, that SAT solvers are not particularly efficient at factorizing large numbers. But if a small factorization problem occurs as part of a larger circuit problem, the Yosys SAT solver is perfectly capable of solving it.

C. Solving sequential SAT problems

The SAT solver functionality in Yosys can not only be used to solve combinatorial problems, but can also solve sequential problems. Let's consider the entire `memdemo` module from Fig. 14 and suppose we want to know which sequence of input values for `d` will cause the output `y` to produce the sequence 1, 2, 3 from any initial state. Fig. 20 show the solution to this question, as produced by the following command:

```
sat -seq 6 -show y -show d -set-init-undef \
  -max_undef -set-at 4 y 1 -set-at 5 y 2 -set-at 6 y 3
```

The `-seq 6` option instructs the `sat` command to solve a sequential problem in 6 time steps. (Experiments with lower number of steps have show that at least 3 cycles are necessary to bring the circuit in a state from which the sequence 1, 2, 3 can be produced.)

```
1  module primetest(p, a, b, ok);
2  input [15:0] p, a, b;
3  output ok = p != a*b || a == 1 || b == 1;
4  endmodule
```

Figure 18. A simple miter circuit for testing if a number is prime. But it has a problem (see main text and Fig. 19).

The `-set-init-undef` option tells the `sat` command to initialize all registers to the undef (`x`) state. The way the `x` state is treated in Verilog will ensure that the solution will work for any initial state.

The `-max_undef` option instructs the `sat` command to find a solution with a maximum number of undefs. This way we can see clearly which inputs bits are relevant to the solution.

Finally the three `-set-at` options add constraints for the `y` signal to play the 1, 2, 3 sequence, starting with time step 4.

It is not surprising that the solution sets `d = 0` in the first step, as this is the only way of setting the `s1` and `s2` registers to a known value. The input values for the other steps are a bit harder to work out manually, but the SAT solver finds the correct solution in an instant.

There is much more to write about the `sat` command. For example, there is a set of options that can be used to performs sequential proofs using temporal induction [6]. The command `help sat` can be used to print a list of all options with short descriptions of their functions.

VI. Conclusion

Yosys provides a wide range of functions to analyze and investigate designs. For many cases it is sufficient to simply display circuit diagrams, maybe use some additional commands to narrow the scope of the circuit diagrams to the interesting parts of the circuit. But some cases require more than that. For this applications Yosys provides commands that can be used to further inspect the behavior of the circuit, either by evaluating which output values are generated from certain input values (`eval`) or by evaluation which input values and initial conditions can result in a certain behavior at the outputs (`sat`). The SAT command can even be used to prove (or disprove) theorems regarding the circuit, in more advanced cases with the additional help of a miter circuit.

This features can be powerful tools for the circuit designer using Yosys as a utility for building circuits and the software developer using Yosys as a framework for new algorithms alike.

References

[1] Clifford Wolf. The Yosys Open SYnthesis Suite. http://www.clifford.at/yosys/
[2] Graphviz - Graph Visualization Software. http://www.graphviz.org/
[3] xdot.py - an interactive viewer for graphs written in Graphviz's dot language. https://github.com/jrfonseca/xdot.py
[4] *Circuit satisfiability problem* on Wikipedia http://en.wikipedia.org/wiki/Circuit_satisfiability
[5] MiniSat: a minimalistic open-source SAT solver. http://minisat.se/
[6] Niklas Een and Niklas Sörensson (2003). Temporal Induction by Incremental SAT Solving. http://citeseerx.ist.psu.edu/viewdoc/summary?doi=10.1.1.4.8161

```
 1 | yosys [primetest]> sat -prove ok 1 -set p 31
 2 |
 3 | 8. Executing SAT pass (solving SAT problems in the circuit).
 4 | Full command line: sat -prove ok 1 -set p 31
 5 |
 6 | Setting up SAT problem:
 7 | Import set-constraint: \p = 16'0000000000011111
 8 | Final constraint equation: \p = 16'0000000000011111
 9 | Imported 6 cells to SAT database.
10 | Import proof-constraint: \ok = 1'1
11 | Final proof equation: \ok = 1'1
12 |
13 | Solving problem with 2790 variables and 8241 clauses..
14 | SAT proof finished - model found: FAIL!
15 |
16 |    _____                 ___      ___       _ _            _ _
17 |   (_____ \               / __)    / __)     (_) |          | | |
18 |    _____) )___ ___ ___ _| |__    _| |__ _____ _| | _____  __| | |
19 |   |  ____/ ___) _ \ / _ (_   __)  (_   __|____ | | || ___ |/ _ | |_|
20 |   | |   | |  | |_| | |_| || |        | | / ___ | | || ___(( (_| |_
21 |   |_|   |_|   \___/ \___/ |_|        |_|  \_____|_|\_)_____)\____|_|
22 |
23 |
24 |    Signal Name              Dec        Hex                  Bin
25 |    ------------------- ---------- ---------- --------------------
26 |    \a                       15029       3ab5        0011101010110101
27 |    \b                        4099       1003        0001000000000011
28 |    \ok                          0          0                       0
29 |    \p                          31         1f        0000000000011111
30 |
31 | yosys [primetest]> sat -prove ok 1 -set p 31 -set a[15:8],b[15:8] 0
32 |
33 | 9. Executing SAT pass (solving SAT problems in the circuit).
34 | Full command line: sat -prove ok 1 -set p 31 -set a[15:8],b[15:8] 0
35 |
36 | Setting up SAT problem:
37 | Import set-constraint: \p = 16'0000000000011111
38 | Import set-constraint: { \a [15:8] \b [15:8] } = 16'0000000000000000
39 | Final constraint equation: { \a [15:8] \b [15:8] \p } = { 16'0000000000000000 16'0000000000011111 }
40 | Imported 6 cells to SAT database.
41 | Import proof-constraint: \ok = 1'1
42 | Final proof equation: \ok = 1'1
43 |
44 | Solving problem with 2790 variables and 8257 clauses..
45 | SAT proof finished - no model found: SUCCESS!
46 |
47 |                 /$$$$$$      /$$$$$$$$     /$$$$$$$
48 |                /$$__  $$    | $$_____/    | $$__  $$
49 |               | $$  \ $$    | $$          | $$  \ $$
50 |               | $$  | $$    | $$$$$       | $$  | $$
51 |               | $$  | $$    | $$__/       | $$  | $$
52 |               | $$/$$ $$    | $$          | $$  | $$
53 |               |  $$$$$$/ /$$| $$$$$$$$ /$$| $$$$$$$//$$
54 |                \____ $$$|__/|_____/|__/|_____/|__/
55 |                     \__/
```

Figure 19. Experiments with the miter circuit from Fig. 18. The first attempt of proving that 31 is prime failed because the SAT solver found a creative way of factorizing 31 using integer overflow.

```
 1 │ yosys [memdemo]> sat -seq 6 -show y -show d -set-init-undef \
 2 │          -max_undef -set-at 4 y 1 -set-at 5 y 2 -set-at 6 y 3
 3 │
 4 │ 6. Executing SAT pass (solving SAT problems in the circuit).
 5 │ Full command line: sat -seq 6 -show y -show d -set-init-undef
 6 │          -max_undef -set-at 4 y 1 -set-at 5 y 2 -set-at 6 y 3
 7 │
 8 │ Setting up time step 1:
 9 │ Final constraint equation: { } = { }
10 │ Imported 29 cells to SAT database.
11 │
12 │ Setting up time step 2:
13 │ Final constraint equation: { } = { }
14 │ Imported 29 cells to SAT database.
15 │
16 │ Setting up time step 3:
17 │ Final constraint equation: { } = { }
18 │ Imported 29 cells to SAT database.
19 │
20 │ Setting up time step 4:
21 │ Import set-constraint for timestep: \y = 4'0001
22 │ Final constraint equation: \y = 4'0001
23 │ Imported 29 cells to SAT database.
24 │
25 │ Setting up time step 5:
26 │ Import set-constraint for timestep: \y = 4'0010
27 │ Final constraint equation: \y = 4'0010
28 │ Imported 29 cells to SAT database.
29 │
30 │ Setting up time step 6:
31 │ Import set-constraint for timestep: \y = 4'0011
32 │ Final constraint equation: \y = 4'0011
33 │ Imported 29 cells to SAT database.
34 │
35 │ Setting up initial state:
36 │ Final constraint equation: { \y \s2 \s1 \mem[3] \mem[2] \mem[1]
37 │                             \mem[0] } = 24'xxxxxxxxxxxxxxxxxxxxxxxx
38 │
39 │ Import show expression: \y
40 │ Import show expression: \d
41 │
42 │ Solving problem with 10322 variables and 27881 clauses..
43 │ SAT model found. maximizing number of undefs.
44 │ SAT solving finished - model found:
45 │
46 │   Time Signal Name                    Dec
   │ Hex││          Bin                                          │
47 │ ││ ---- -------------------- ---------- ---------- ----┼----------
48 │ ││  init \mem[0]                        --          │
   │ -- ││          xxxx                                 │
49 │ ││  init \mem[1]                        --          │
   │ -- ││          xxxx                                 │
50 │ ││  init \mem[2]                        --          │
   │ -- ││          xxxx                                 │
51 │ ││  init \mem[3]                        --          │
   │ -- ││          xxxx                                 │
52 │ ││  init \s1                            --          │
   │ -- ││          xx                                   │
53 │ ││  init \s2                            --          │
   │ -- ││          xx                                   │
54 │ ││  init \y                             --          │
   │ -- ││          xxxx                                 │
55 │ ││ ---- -------------------- ---------- ---------- ----┼----------
56 │ ││     1 \d                             0           │
   │ 0  ││          0000                                 │
57 │ ││     1 \y                             --          │
   │ -- ││          xxxx                                 │
58 │ ││ ---- -------------------- ---------- ---------- ----┼----------
59 │ ││     2 \d                             1           │
   │ 1  ││          0001                                 │
60 │ ││     2 \y                             --          │
   │ -- ││          xxxx                                 │
61 │ ││ ---- -------------------- ---------- ---------- ----┼----------
62 │ ││     3 \d                             2           │
   │ 2  ││          0010                                 │
63 │ ││     3 \y                             0           │
   │ 0  ││          0000                                 │
64 │ ││ ---- -------------------- ---------- ---------- ----┼----------
65 │ ││     4 \d                             3           │
   │ 3  ││          0011                                 │
66 │ ││     4 \y                             1           │
   │ 1  ││          0001                                 │
67 │ ││ ---- -------------------- ---------- ---------- ----┼----------
68 │ ││     5 \d                             --          │
   │ -- ││          001x                                 │
69 │ ││     5 \y                             2           │
   │ 2  ││          0010                                 │
70 │ ││ ---- -------------------- ---------- ---------- ----┼----────
71 │ ││     6 \d                             --         170
   │ -- ││          xxxx                                 │
72 │ ││     6 \y                             3           │
   │ 3  ││          0011                                 │
```

Figure 20. Solving a sequential SAT problem in the memdemo module from
Fig. 14.

Yosys Application Note 012:
Converting Verilog to BTOR

Ahmed Irfan and Clifford Wolf

April 2015

Abstract—**Verilog-2005 is a powerful Hardware Description Language (HDL) that can be used to easily create complex designs from small HDL code. BTOR [3] is a bit-precise word-level format for model checking. It is a simple format and easy to parse. It allows to model the model checking problem over the theory of bit-vectors with one-dimensional arrays, thus enabling to model Verilog designs with registers and memories. Yosys [1] is an Open-Source Verilog synthesis tool that can be used to convert Verilog designs with simple assertions to BTOR format.**

I. INSTALLATION

Yosys written in C++ (using features from C++11) and is tested on modern Linux. It should compile fine on most UNIX systems with a C++11 compiler. The README file contains useful information on building Yosys and its prerequisites.

Yosys is a large and feature-rich program with some dependencies. For this work, we may deactivate other extra features such as TCL and ABC support in the `Makefile`.

This Application Note is based on GIT Rev. `082550f` from 2015-04-04 of Yosys [1].

II. QUICK START

We assume that the Verilog design is synthesizable and we also assume that the design does not have multi-dimensional memories. As BTOR implicitly initializes registers to zero value and memories stay uninitialized, we assume that the Verilog design does not contain initial blocks. For more details about the BTOR format, please refer to [3].

We provide a shell script `verilog2btor.sh` which can be used to convert a Verilog design to BTOR. The script can be found in the `backends/btor` directory. The following example shows its usage:

```
verilog2btor.sh fsm.v fsm.btor test
```

Listing 1. Using verilog2btor script

The script `verilog2btor.sh` takes three parameters. In the above example, the first parameter `fsm.v` is the input design, the second parameter `fsm.btor` is the file name of BTOR output, and the third parameter `test` is the name of top module in the design.

To specify the properties (that need to be checked), we have two options:

- We can use the Verilog `assert` statement in the procedural block or module body of the Verilog design, as shown in Listing 2. This is the preferred option.
- We can use a single-bit output wire, whose name starts with `safety`. The value of this output wire needs to be driven low when the property is met, i.e. the solver will try to find a model that makes the safety pin go high. This is demonstrated in Listing 3.

```verilog
module test(input clk, input rst, output y);

  reg [2:0] state;

  always @(posedge clk) begin
    if (rst || state == 3) begin
      state <= 0;
    end else begin
      assert(state < 3);
      state <= state + 1;
    end
  end

  assign y = state[2];

  assert property (y !== 1'b1);

endmodule
```

Listing 2. Specifying property in Verilog design with `assert`

```verilog
module test(input clk, input rst,
    output y, output safety1);

  reg [2:0] state;

  always @(posedge clk) begin
    if (rst || state == 3)
      state <= 0;
    else
      state <= state + 1;
  end

  assign y = state[2];

  assign safety1 = !(y !== 1'b1);

endmodule
```

Listing 3. Specifying property in Verilog design with output wire

We can run Boolector [2] 1.4.1[1] on the generated BTOR file:

```
$ boolector fsm.btor
unsat
```

Listing 4. Running boolector on BTOR file

We can also use nuXmv [4], but on BTOR designs it does not support memories yet. With the next release of nuXmv, we will be also able to verify designs with memories.

III. DETAILED FLOW

Yosys is able to synthesize Verilog designs up to the gate level. We are interested in keeping registers and memories when synthesizing the design. For this purpose, we describe a customized Yosys synthesis flow, that is also provided by the `verilog2btor.sh`

---

[1] Newer version of Boolector do not support sequential models. Boolector 1.4.1 can be built with picosat-951. Newer versions of picosat have an incompatible API.

script. Listing 5 shows the Yosys commands that are executed by `verilog2btor.sh`.

```
1  read_verilog -sv $1;
2  hierarchy -top $3; hierarchy -libdir $DIR;
3  hierarchy -check;
4  proc; opt;
5  opt_expr -mux_undef; opt;
6  rename -hide;;;
7  splice; opt;
8  memory_dff -wr_only; memory_collect;;
9  flatten;;
10 memory_unpack;
11 splitnets -driver;
12 setundef -zero -undriven;
13 opt;;;
14 write_btor $2;
```

Listing 5. Synthesis Flow for BTOR with memories

Here is short description of what is happening in the script line by line:

1) Reading the input file.
2) Setting the top module in the hierarchy and trying to read automatically the files which are given as `include` in the file read in first line.
3) Checking the design hierarchy.
4) Converting processes to multiplexers (muxs) and flip-flops.
5) Removing undef signals from muxs.
6) Hiding all signal names that are not used as module ports.
7) Explicit type conversion, by introducing slice and concat cells in the circuit.
8) Converting write memories to synchronous memories, and collecting the memories to multi-port memories.
9) Flattening the design to get only one module.
10) Separating read and write memories.
11) Splitting the signals that are partially assigned
12) Setting undef to zero value.
13) Final optimization pass.
14) Writing BTOR file.

For detailed description of the commands mentioned above, please refer to the Yosys documentation, or run `yosys -h` *command_name*.

The script presented earlier can be easily modified to have a BTOR file that does not contain memories. This is done by removing the line number 8 and 10, and introduces a new command `memory` at line number 8. Listing 6 shows the modified Yosys script file:

```
read_verilog -sv $1;
hierarchy -top $3; hierarchy -libdir $DIR;
hierarchy -check;
proc; opt;
opt_expr -mux_undef; opt;
rename -hide;;;
splice; opt;
memory;;
flatten;;
splitnets -driver;
setundef -zero -undriven;
opt;;;
write_btor $2;
```

Listing 6. Synthesis Flow for BTOR without memories

IV. EXAMPLE

Here is an example Verilog design that we want to convert to BTOR:

```verilog
module array(input clk);

  reg [7:0] counter;
  reg [7:0] mem [7:0];

  always @(posedge clk) begin
    counter <= counter + 8'd1;
    mem[counter] <= counter;
  end

  assert property (!(counter > 8'd0) ||
    mem[counter - 8'd1] == counter - 8'd1);

endmodule
```

Listing 7. Example - Verilog Design

The generated BTOR file that contain memories, using the script shown in Listing 5:

```
1 var 1 clk
2 array 8 3
3 var 8 $auto$rename.cc:150:execute$20
4 const 8 00000001
5 sub 8 3 4
6 slice 3 5 2 0
7 read 8 2 6
8 slice 3 3 2 0
9 add 8 3 4
10 const 8 00000000
11 ugt 1 3 10
12 not 1 11
13 const 8 11111111
14 slice 1 13 0 0
15 one 1
16 eq 1 1 15
17 and 1 16 14
18 write 8 3 2 8 3
19 acond 8 3 17 18 2
20 anext 8 3 2 19
21 eq 1 7 5
22 or 1 12 21
23 const 1 1
24 one 1
25 eq 1 23 24
26 cond 1 25 22 24
27 root 1 -26
28 cond 8 1 9 3
29 next 8 3 28
```

Listing 8. Example - Converted BTOR with memory

```
1 var 1 clk
2 var 8 mem[0]
3 var 8 $auto$rename.cc:150:execute$20
4 slice 3 3 2 0
5 slice 1 4 0 0
6 not 1 5
7 slice 1 4 1 1
8 not 1 7
9 slice 1 4 2 2
10 not 1 9
11 and 1 8 10
12 and 1 6 11
13 cond 8 12 3 2
14 cond 8 1 13 2
15 next 8 2 14
16 const 8 00000001
17 add 8 3 16
18 const 8 00000000
19 ugt 1 3 18
20 not 1 19
21 var 8 mem[2]
22 and 1 7 10
23 and 1 6 22
24 cond 8 23 3 21
25 cond 8 1 24 21
26 next 8 21 25
27 sub 8 3 16
  ⋮
54 cond 1 53 50 52
55 root 1 -54
  ⋮
77 cond 8 76 3 44
78 cond 8 1 77 44
79 next 8 44 78
```

Listing 9. Example - Converted BTOR without memory

V. LIMITATIONS

BTOR does not support initialization of memories and registers, i.e. they are implicitly initialized to value zero, so the initial block for memories need to be removed when converting to BTOR. It should also be kept in consideration that BTOR does not support the x or z values of Verilog.

Another thing to bear in mind is that Yosys will convert multi-dimensional memories to one-dimensional memories and address decoders. Therefore out-of-bounds memory accesses can yield unexpected results.

VI. CONCLUSION

Using the described flow, we can use Yosys to generate word-level verification benchmarks with or without memories from Verilog designs.

REFERENCES

[1] Clifford Wolf. The Yosys Open SYnthesis Suite.
    http://www.clifford.at/yosys/
[2] Robert Brummayer and Armin Biere, Boolector: An Efficient SMT Solver for Bit-Vectors and Arrays
    http://fmv.jku.at/boolector/

And the BTOR file obtained by the script shown in Listing 6, which expands the memory into individual elements:

[3] Robert Brummayer and Armin Biere and Florian Lonsing, BTOR: Bit-Precise Modelling of Word-Level Problems for Model Checking
http://fmv.jku.at/papers/BrummayerBiereLonsing-BPR08.pdf

[4] Roberto Cavada and Alessandro Cimatti and Michele Dorigatti and Alberto Griggio and Alessandro Mariotti and Andrea Micheli and Sergio Mover and Marco Roveri and Stefano Tonetta, The nuXmv Symbolic Model Checker
https://es-static.fbk.eu/tools/nuxmv/index.php

Bibliography

[ASU86]   Aho, Alfred V. ; Sethi, Ravi ; Ullman, Jeffrey D.: *Compilers: principles, techniques, and tools.* Boston, MA, USA : Addison-Wesley Longman Publishing Co., Inc., 1986. – ISBN 0–201–10088–6

[BHSV90] Brayton, R.K. ; Hachtel, G.D. ; Sangiovanni-Vincentelli, A.L.: Multilevel logic synthesis. In: *Proceedings of the IEEE* 78 (1990), Nr. 2, S. 264–300. http://dx.doi.org/10.1109/5.52213. – DOI 10.1109/5.52213. – ISSN 0018–9219

[CI00]   Cummings, Clifford E. ; Inc, Sunburst D.: Nonblocking Assignments in Verilog Synthesis, Coding Styles That Kill. In: *SNUG (Synopsys Users Group) 2000 User Papers, section-MC1 (1 st paper,* 2000

[GW13]   Glaser, Johann ; Wolf, Clifford: Methodology and Example-Driven Interconnect Synthesis for Designing Heterogeneous Coarse-Grain Reconfigurable Architectures. In: Haase, Jan (Hrsg.): *Advances in Models, Methods, and Tools for Complex Chip Design — Selected contributions from FDL'12.* Springer, 2013. – to appear

[HS96]   Hachtel, G D. ; Somenzi, F: *Logic Synthesis and Verification Algorithms.* 1996

[IP-10]   IEEE Standard for IP-XACT, Standard Structure for Packaging, Integrating, and Reusing IP within Tools Flows. In: *IEEE Std 1685-2009* (2010), S. C1–360. http://dx.doi.org/10.1109/IEEESTD.2010.5417309. – DOI 10.1109/IEEESTD.2010.5417309

[LHBB85] Lee, Kyu Y. ; Holley, Michael ; Bailey, Mary ; Bright, Walter: A High-Level Design Language for Programmable Logic Devices. In: *VLSI Design (Manhasset NY: CPM Publications)* (June 1985), S. 50–62

[STGR10] Shi, Yiqiong ; Ting, Chan W. ; Gwee, Bah-Hwee ; Ren, Ye: A highly efficient method for extracting FSMs from flattened gate-level netlist. In: *Circuits and Systems (ISCAS), Proceedings of 2010 IEEE International Symposium on,* 2010, S. 2610–2613

[Ull76]   Ullmann, J. R.: An Algorithm for Subgraph Isomorphism. In: *J. ACM* 23 (1976), Januar, Nr. 1, S. 31–42. http://dx.doi.org/10.1145/321921.321925. – DOI 10.1145/321921.321925. – ISSN 0004–5411

[Ver02]   IEEE Standard for Verilog Register Transfer Level Synthesis. In: *IEEE Std 1364.1-2002* (2002). http://dx.doi.org/10.1109/IEEESTD.2002.94220. – DOI 10.1109/IEEESTD.2002.94220

[Ver06]   IEEE Standard for Verilog Hardware Description Language. In: *IEEE Std 1364-2005 (Revision of IEEE Std 1364-2001)* (2006). http://dx.doi.org/10.1109/IEEESTD.2006.99495. – DOI 10.1109/IEEESTD.2006.99495

[VHD04]   IEEE Standard for VHDL Register Transfer Level (RTL) Synthesis. In: *IEEE Std 1076.6-2004 (Revision of IEEE Std 1076.6-1999)* (2004). http://dx.doi.org/10.1109/IEEESTD.2004.94802. – DOI 10.1109/IEEESTD.2004.94802

[VHD09]   IEEE Standard VHDL Language Reference Manual. In: *IEEE Std 1076-2008 (Revision of IEEE Std 1076-2002)* (2009), 26. http://dx.doi.org/10.1109/IEEESTD.2009.4772740. – DOI 10.1109/IEEESTD.2009.4772740

[WGS⁺12] Wolf, Clifford ; Glaser, Johann ; Schupfer, Florian ; Haase, Jan ; Grimm, Christoph: Example-driven interconnect synthesis for heterogeneous coarse-grain reconfigurable logic. In: *FDL Proceeding of the 2012 Forum on Specification and Design Languages*, 2012, S. 194–201

[Wol13] Wolf, Clifford: *Design and Implementation of the Yosys Open SYnthesis Suite.* 2013. – Bachelor Thesis, Vienna University of Technology

Internet References

[16] C-to-Verilog. http://www.c-to-verilog.com/.

[17] Flex. http://flex.sourceforge.net/.

[18] GNU Bison. http://www.gnu.org/software/bison/.

[19] LegUp. http://legup.eecg.utoronto.ca/.

[20] OpenCores I^{2}C Core. http://opencores.org/project,i2c.

[21] OpenCores k68 Core. http://opencores.org/project,k68.

[22] openMSP430 CPU. http://opencores.org/project,openmsp430.

[23] OpenRISC 1200 CPU. http://opencores.org/or1k/OR1200_OpenRISC_Processor.

[24] Synopsys Formality Equivalence Checking. http://www.synopsys.com/Tools/Verification/FormalEquivalence/Pages/Formality.aspx.

[25] The Liberty Library Modeling Standard. http://www.opensourceliberty.org/.

[26] Armin Biere, Johannes Kepler University Linz, Austria. AIGER. http://fmv.jku.at/aiger/.

[27] Berkeley Logic Synthesis and Verification Group. ABC: A System for Sequential Synthesis and Verification. HQ Rev b5750272659f, 2012-10-28, http://www.eecs.berkeley.edu/~alanmi/abc/.

[28] M. group at Berkeley studies logic synthesis and verification for VLSI design. MVSIS: Logic Synthesis and Verification. Version 3.0, http://embedded.eecs.berkeley.edu/mvsis/.

[29] M. McCutchen. C++ Big Integer Library. http://mattmccutchen.net/bigint/.