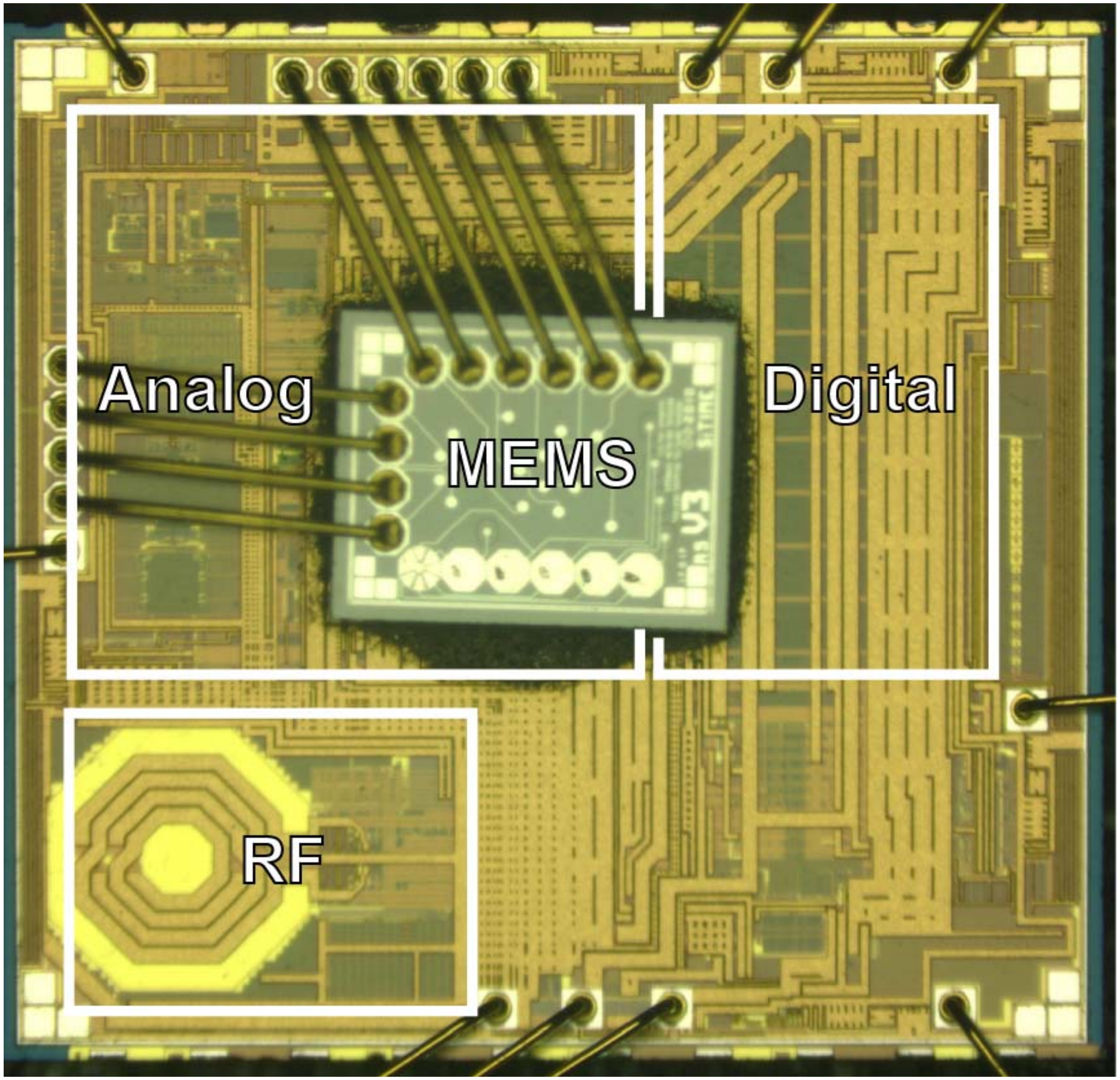
Fast and Accurate System Level Simulation of Time-Based Circuits Using CppSim and VppSim

Michael H. Perrott

March 12, 2016

Modern Mixed Signal Circuit Design

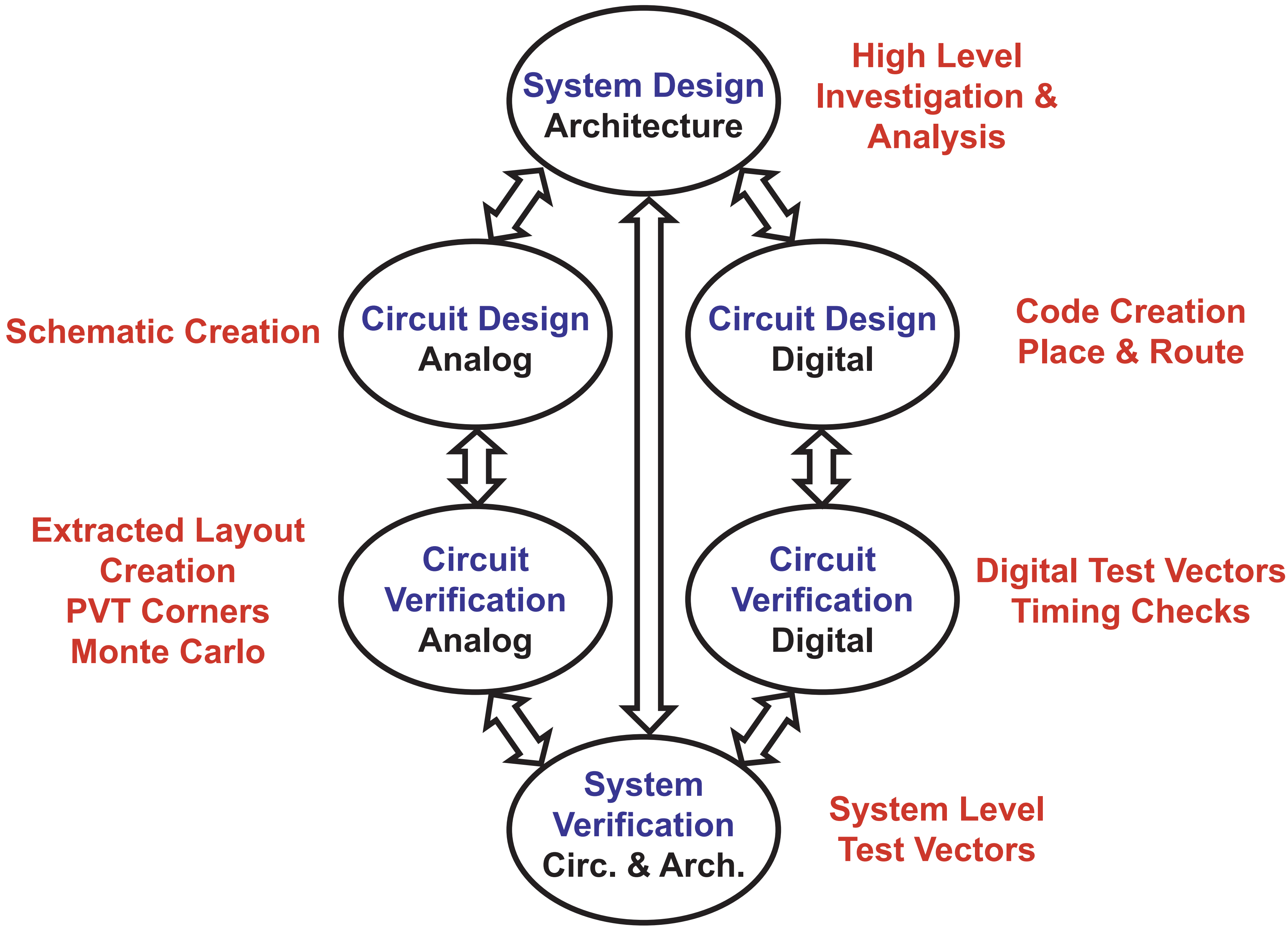


A Programmable MEMS Oscillator

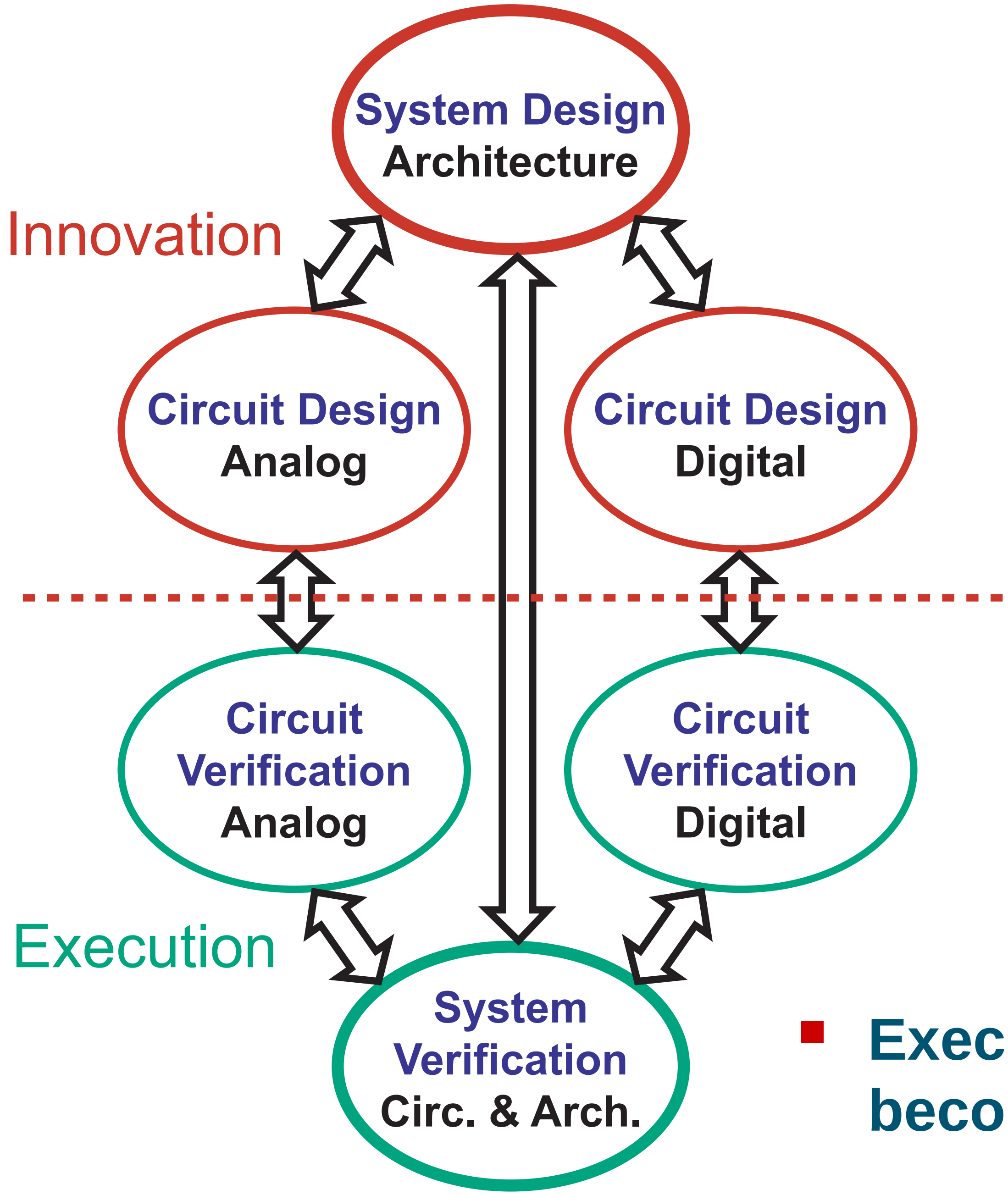
- ***Analog***
  Temperature sensor, ADC, oscillator sustaining circuit
- ***Digital***
  signal processing
- ***RF***
  clocking (2.5 GHz)
- ***MEMS***
  high Q resonator

System level design is critical

Consider a Top Down, Mixed-Signal Design Flow

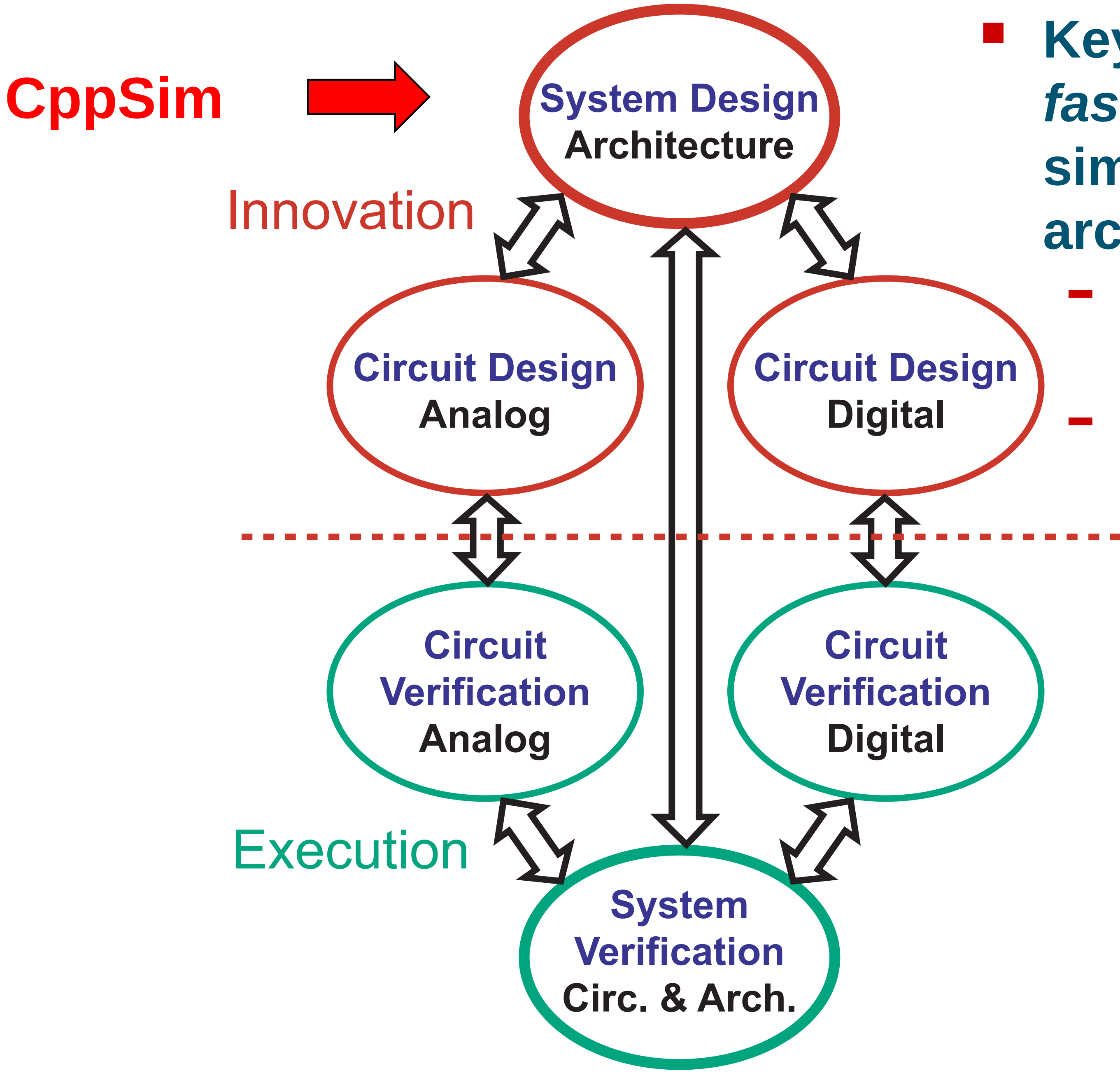


Good Execution Is Certainly A Key to Success



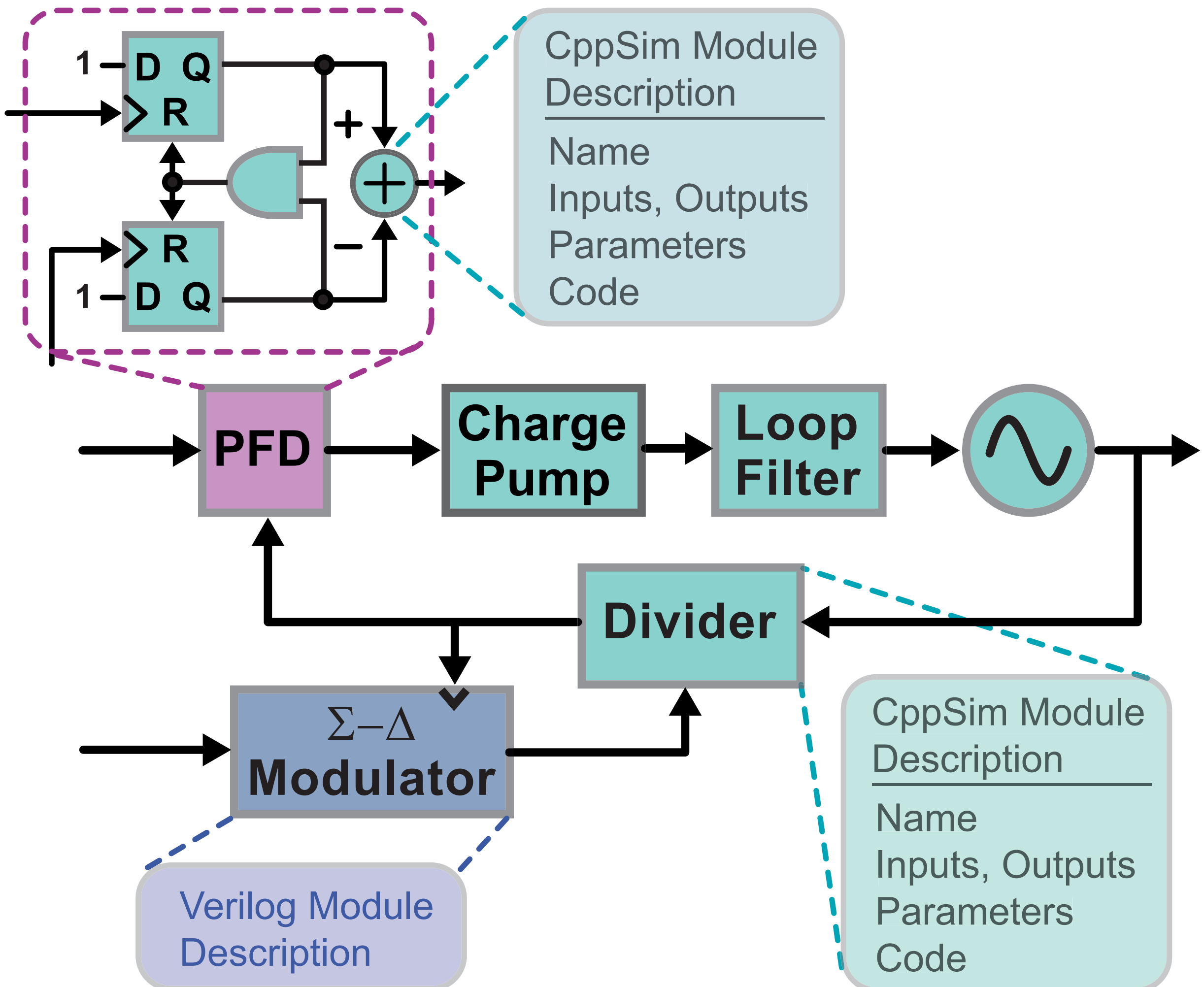
- **Execution often becomes key focus**

New Circuit Architectures Require Innovation



- **Key to innovation is *fast* and *detailed* simulation of new architectures**
  - **Allows evaluation of *many* new ideas**
  - **Pinpoints key problem areas**

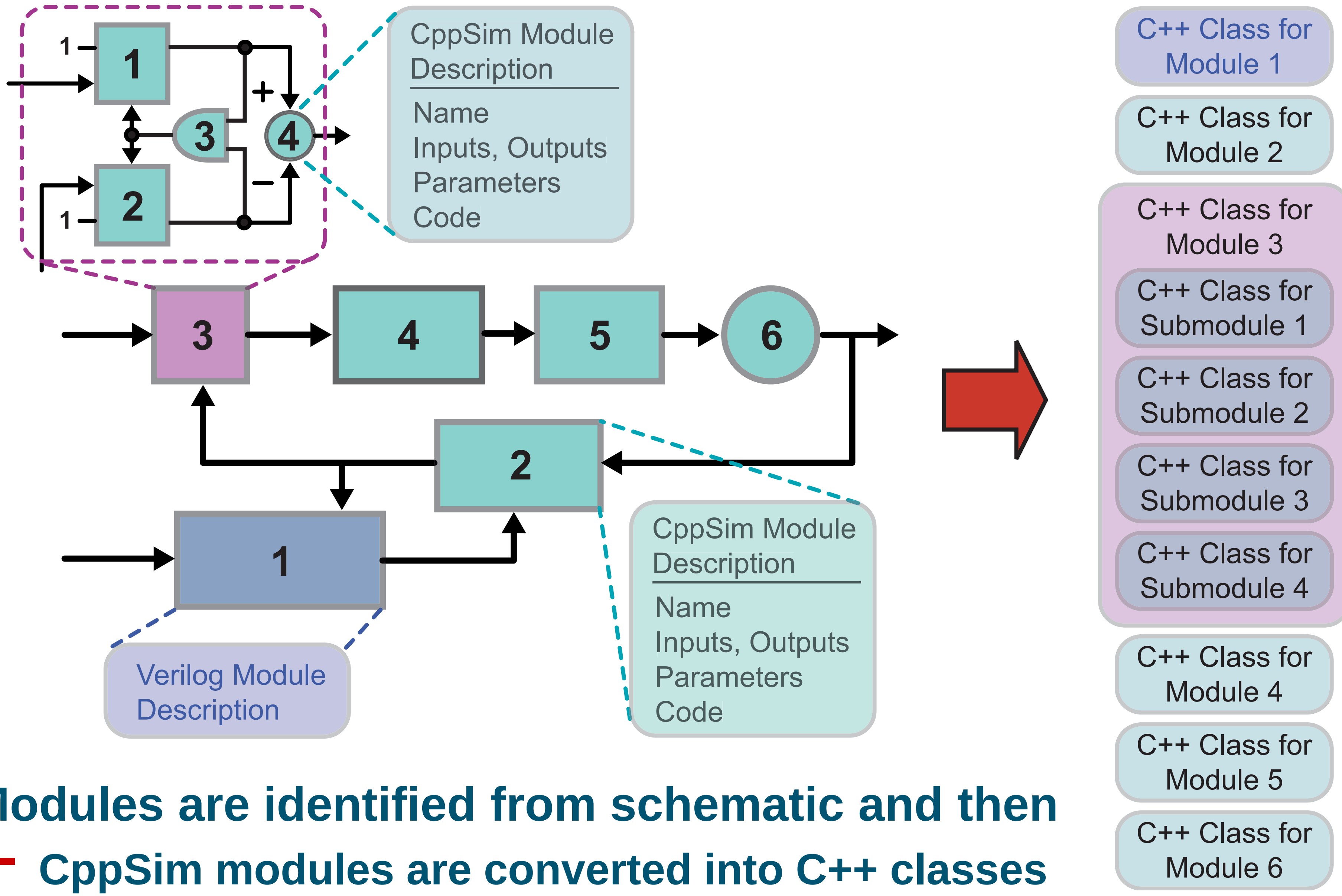
Schematic Based Simulation using CppSim/VppSim



- **Schematic**
  - **Provides hierarchical description of *system topology***
- **Code blocks**
  - **Specify *module behavior* using templated C++ code or Verilog code**

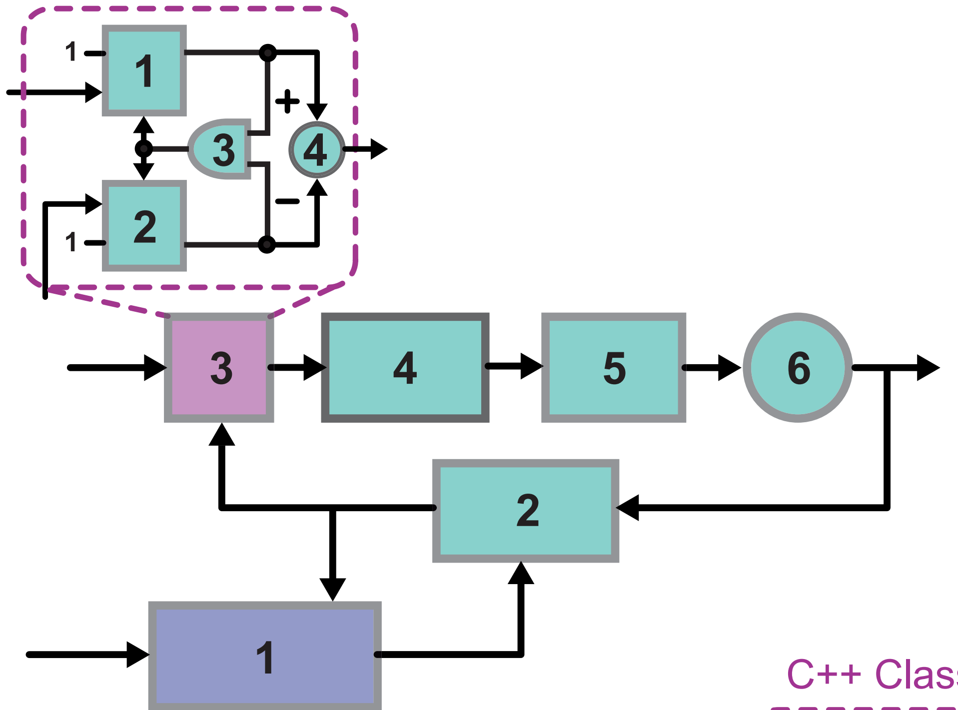
- **Designers graphically develop system based on a library of C++/Verilog symbols and code**
  - **Easy to create new symbols with accompanying code**

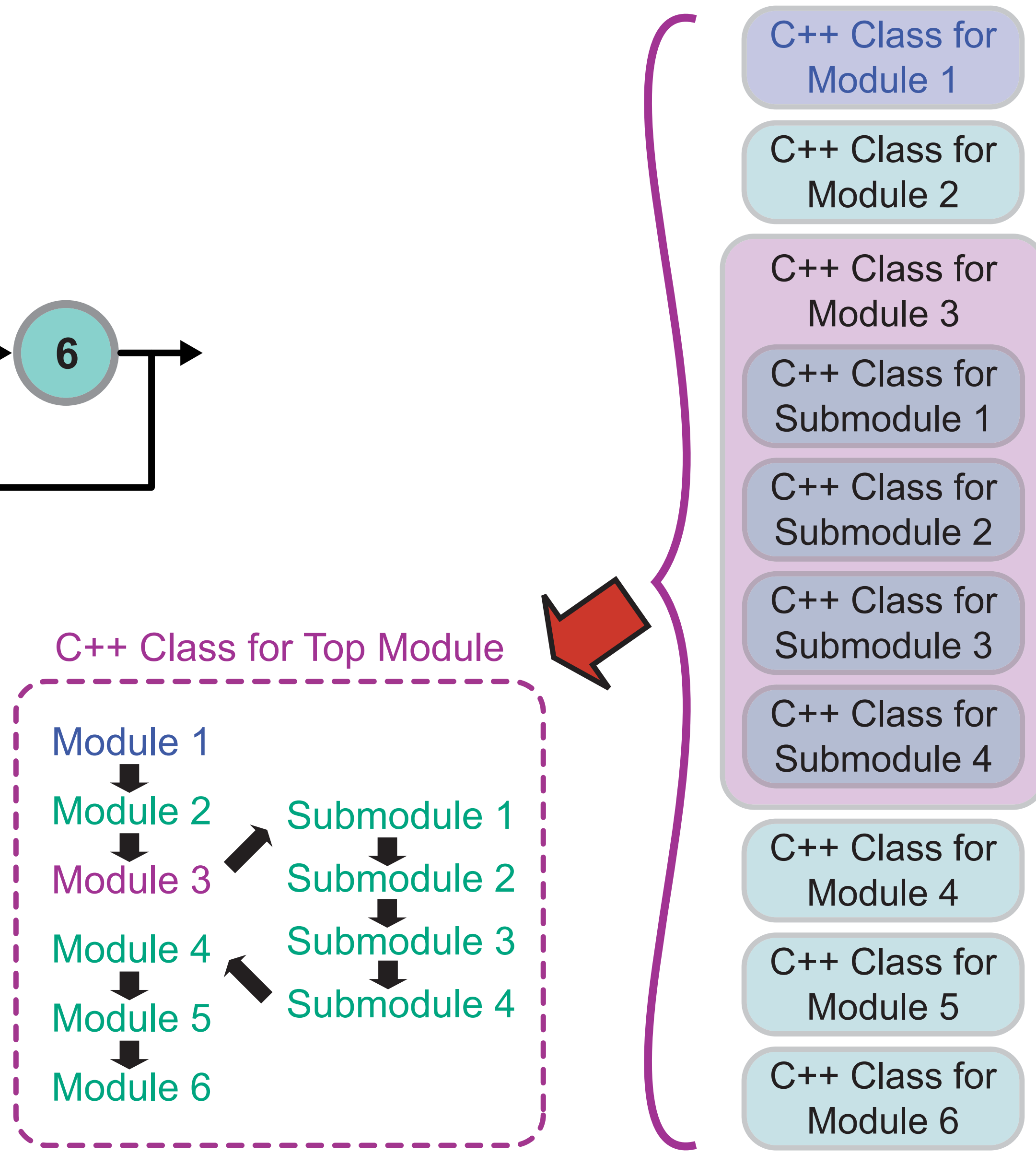
CppSim Automates C++ Class Generation



- **Modules are identified from schematic and then**
  - **CppSim modules are converted into C++ classes**
  - **Verilog modules are translated into C++ classes using Verilator**

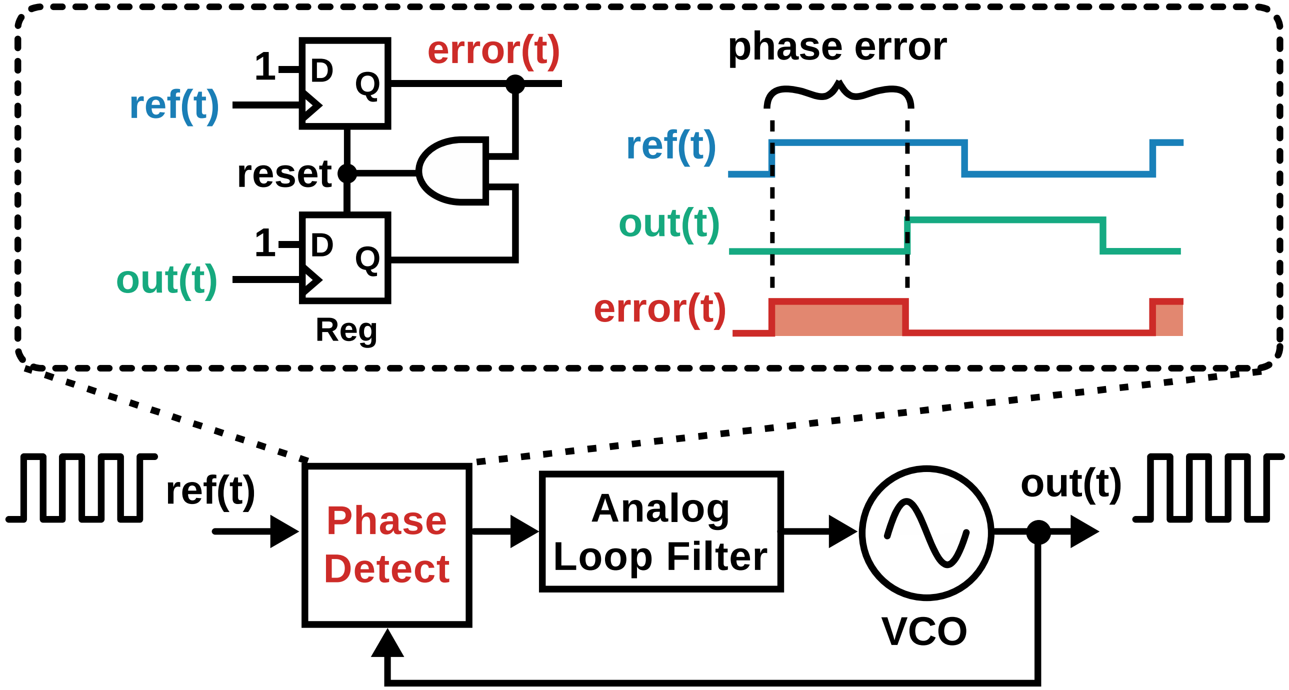
CppSim Assembles C++ Classes into Overall Sim Code



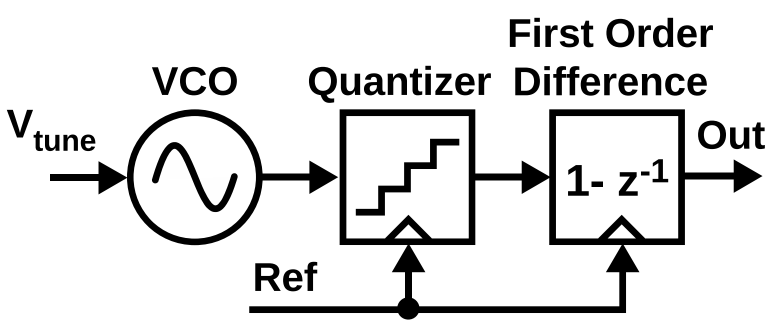


- **Block-by-block execution of each module at each time step**
- **Hierarchical description is retained**

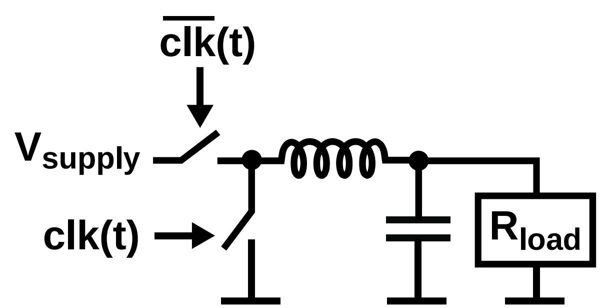
Time As A Signal



Phase-Locked Loops



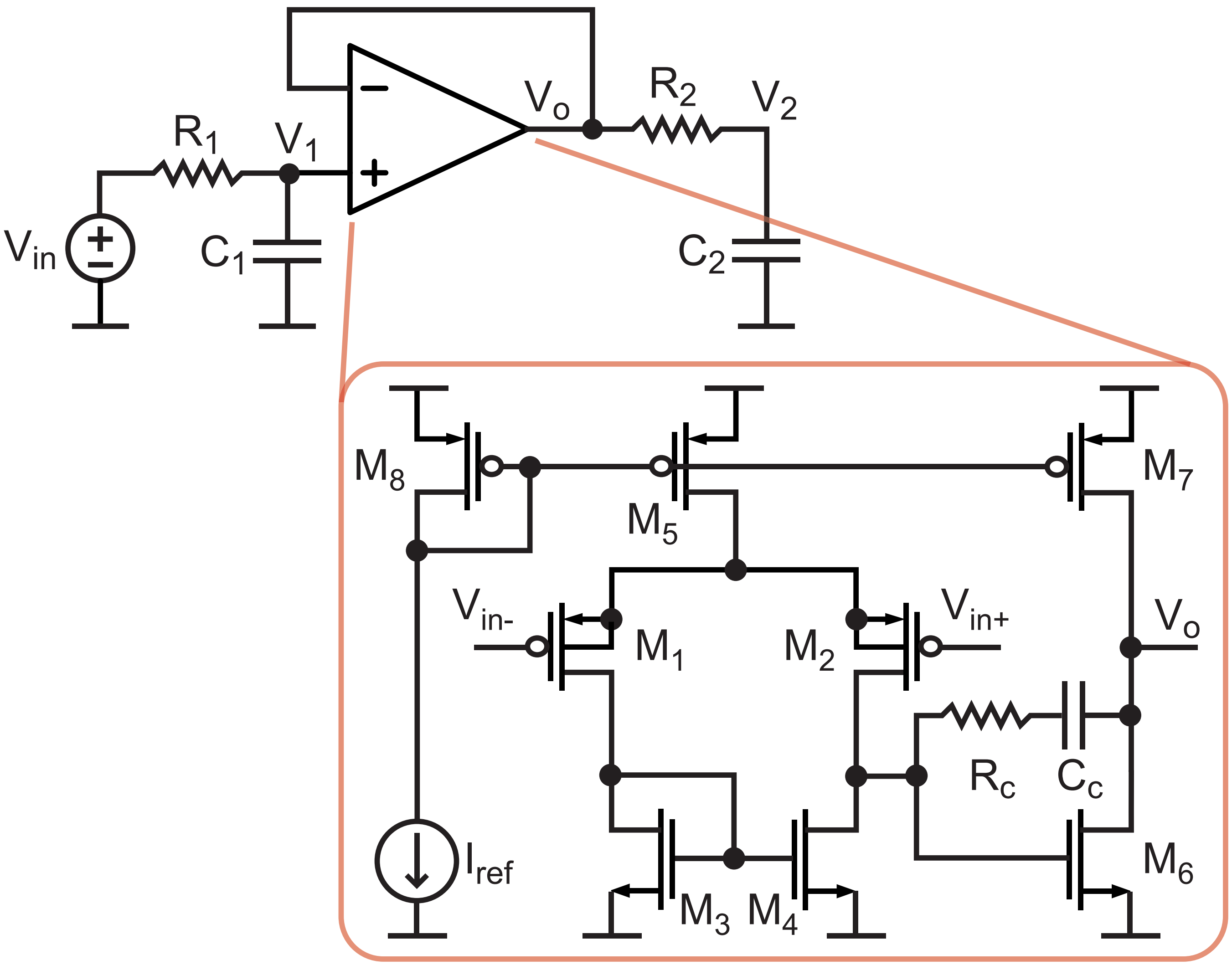
VCO-Based ADCs



Power Conversion Circuits

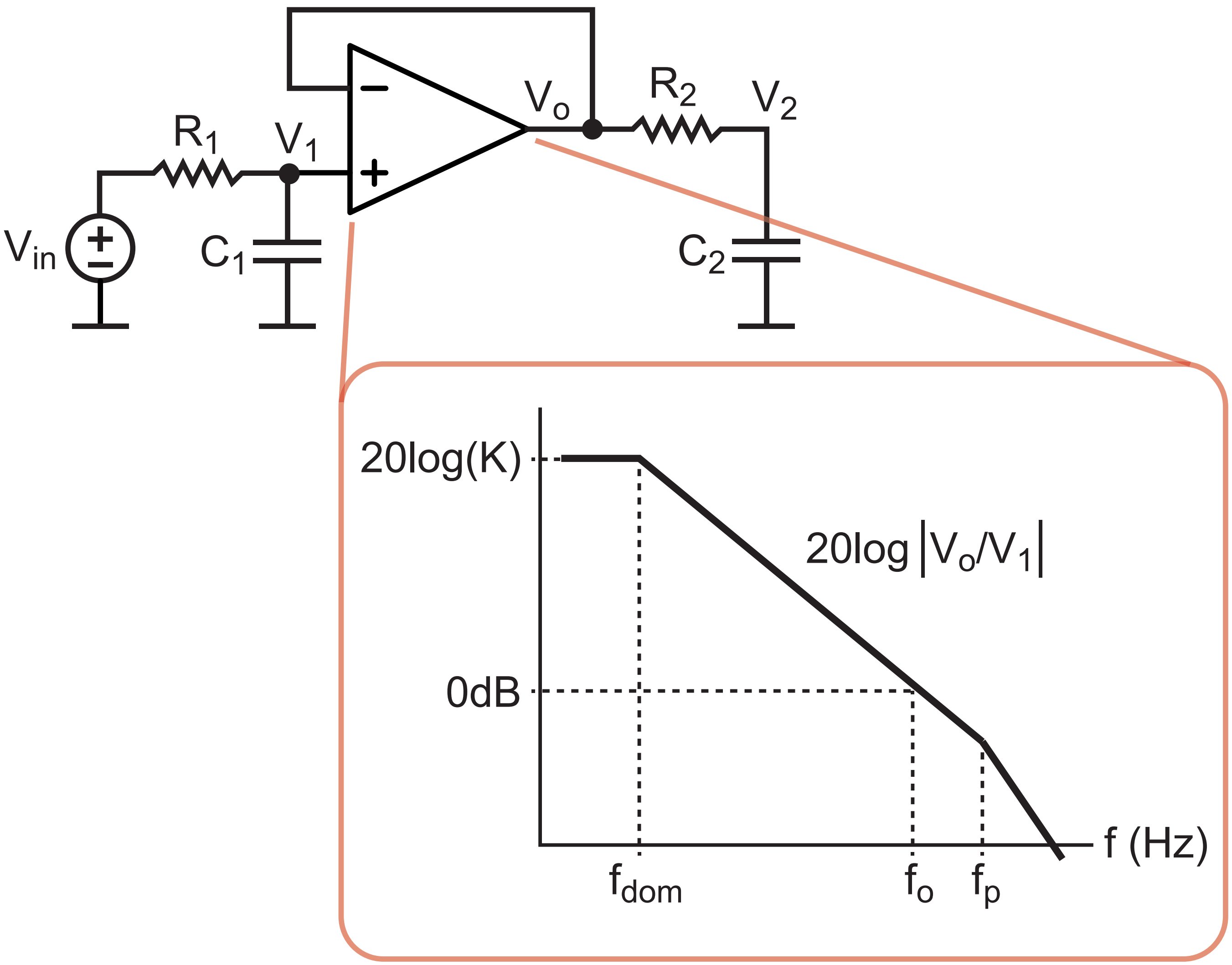
- **CppSim developed to accurately model time in circuits**

System-Level Modeling: A Basic Example



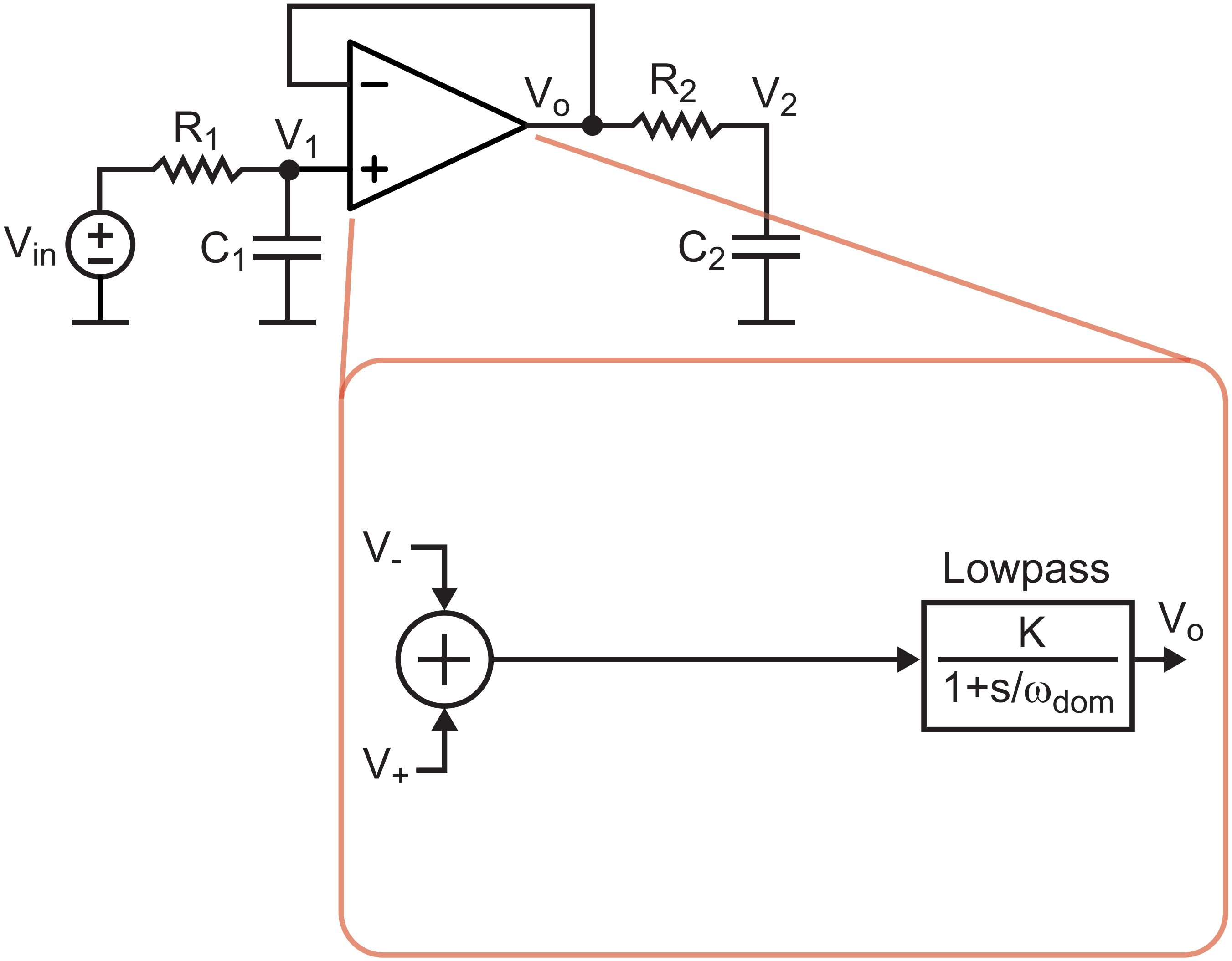
- **Opamp is a nonlinear, transistor-level circuit**
  - **Device level representation mandates SPICE-level simulation**

Opamps Often Modeled at Transfer Function Level



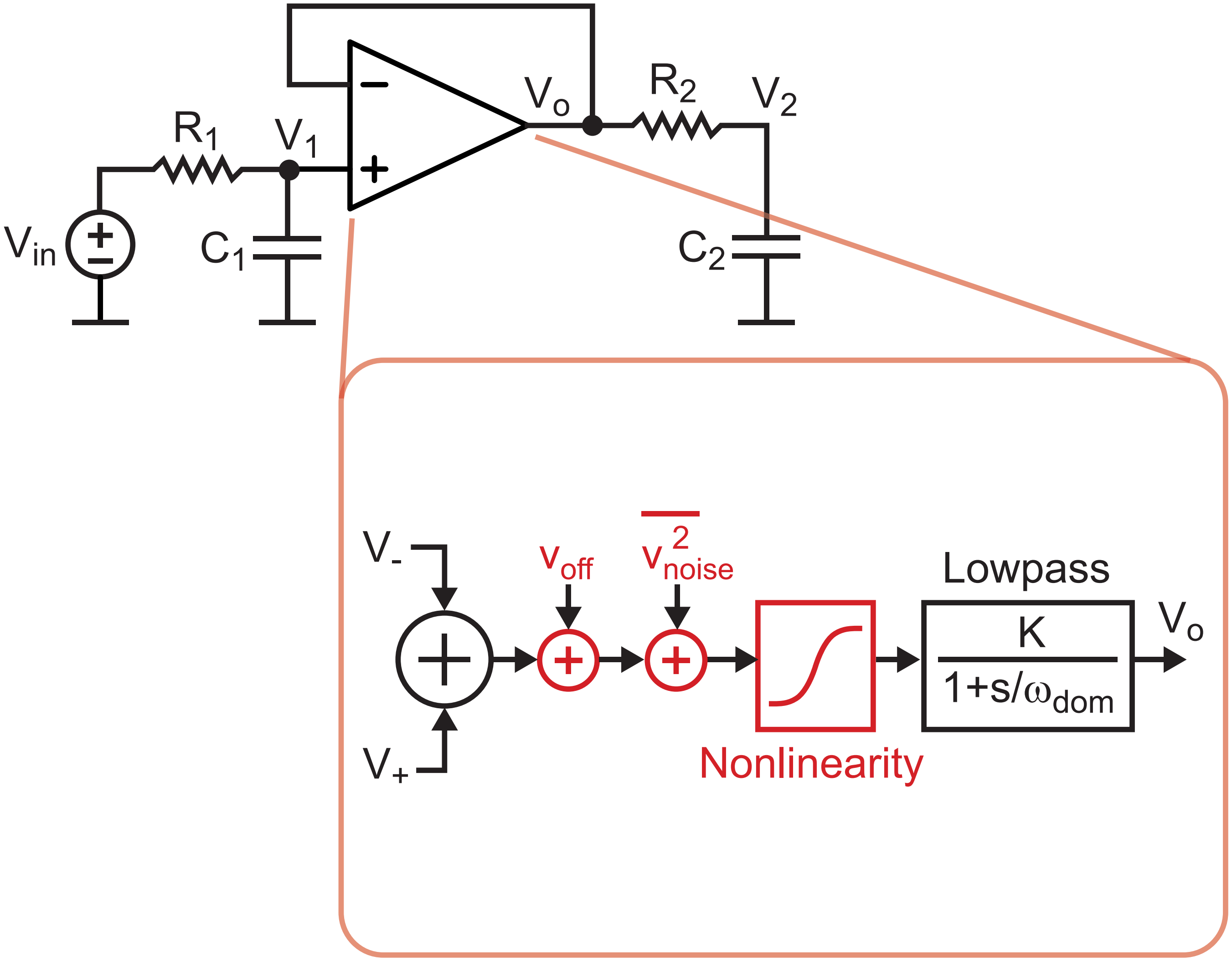
- **Works well for small perturbations about steady-state**
  - **Key parameters are gain and bandwidth**

A Simple Block Diagram Model of Opamp



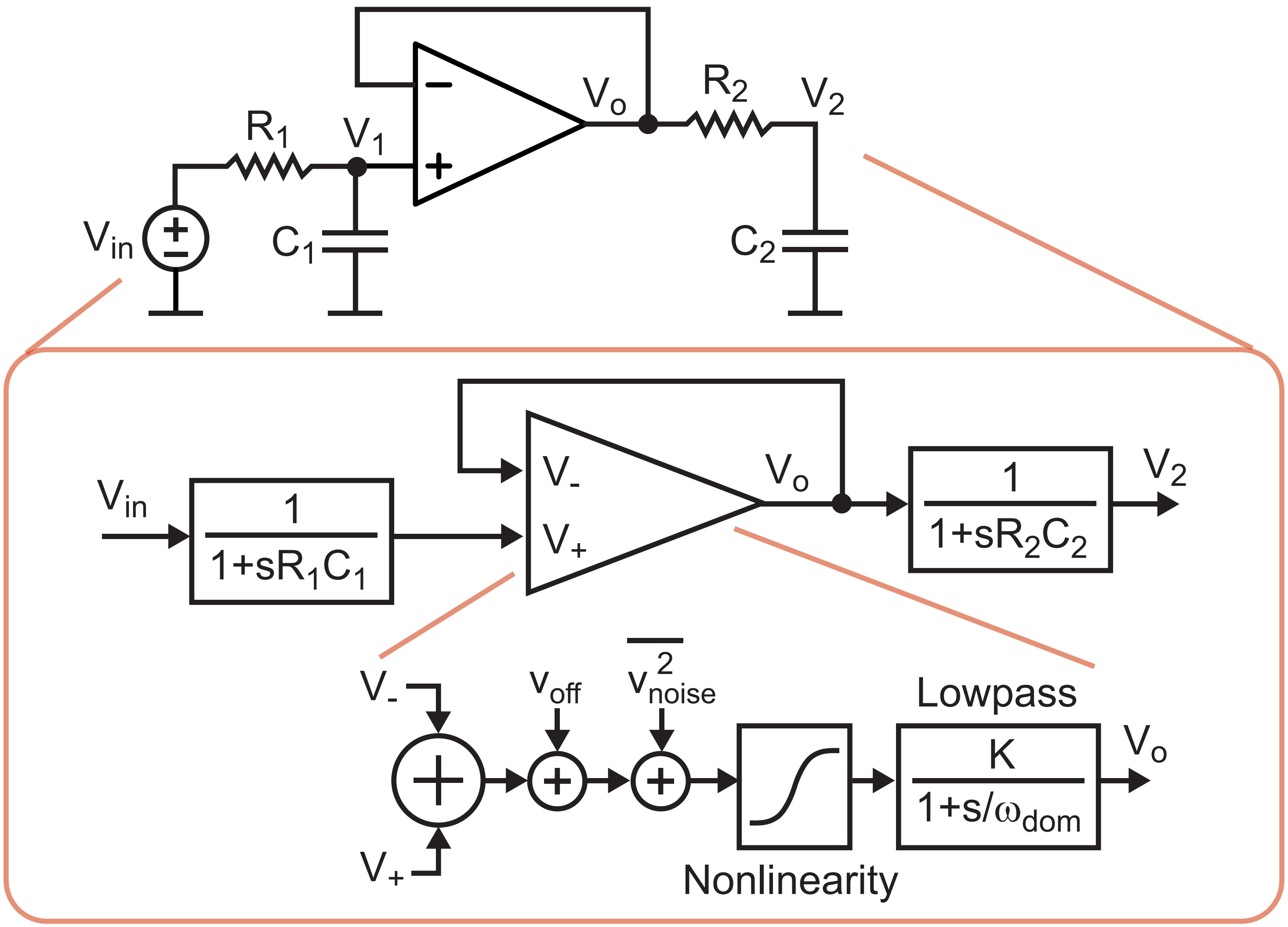
- **Approximates first order behavior of opamp**

Inclusion of Second Order Effects



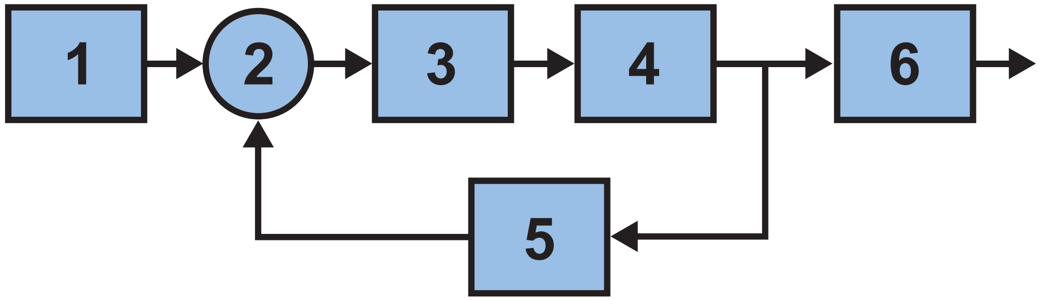
- **Offset, noise, and nonlinearity of front end-differential pair**
  - **Parasitic poles are also easy to add as additional blocks**

Overall Block Diagram Model



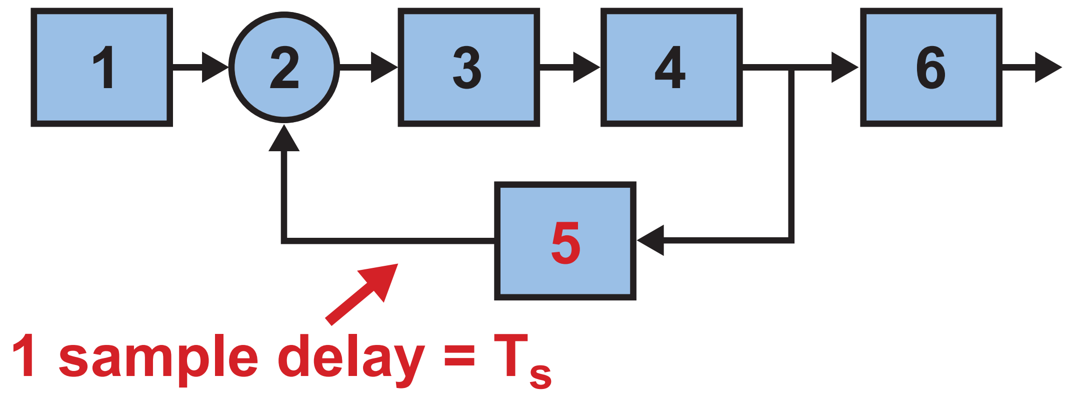
- **Unilateral flow through blocks allows fast simulation**
  - **Compute block outputs one at a time for each time step**

Advantages of Block-by-Block Computation



- **Simple, fast computational structure**
  - **Simply perform computation for each block one at a time for each time step**
    - Extends to hierarchical design quite easily
- **High level of system complexity can be handled**
  - **Overall computational load is simply the sum of the computation required for each block**
  - **Contrast with SPICE whose computational load grows exponentially with the number of elements**

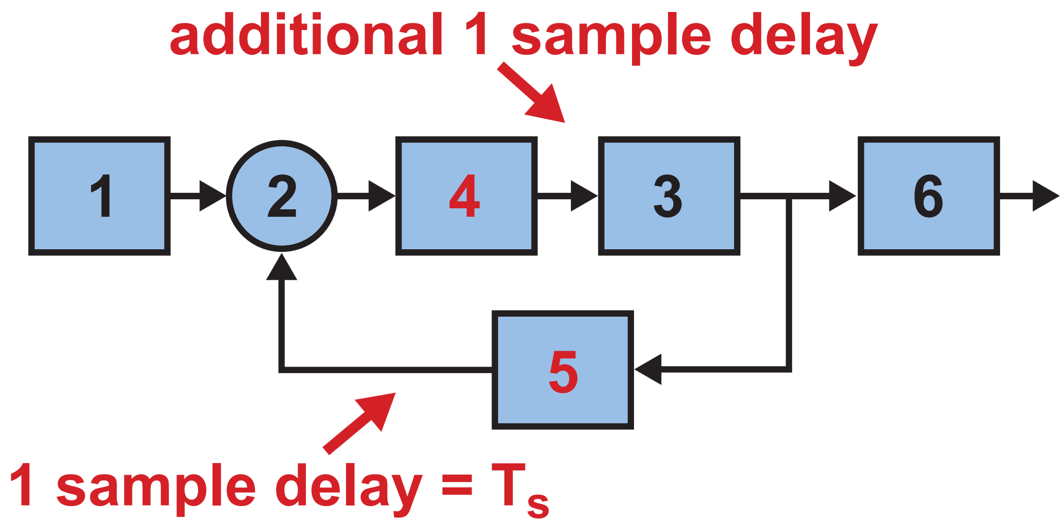
The Issue of Delay with Block-by-Block Computation



- Minimum possible delay within a feedback loop is one sample period
  - Example: Block 2 will not receive updated value from Block 5 until next time sample
  - For unity gain crossover frequency f_o and delay T_s:
    - Phase margin reduced by $f_o \bullet T_s \bullet 360^o$

Time step of simulation must be small compared to bandwidth of feedback loops being simulated

The Issue of Block Order

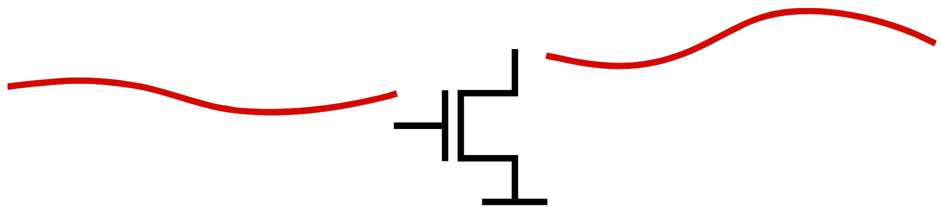


- **Poor ordering of blocks leads to additional delay within feedback loops**
  - **Issue is made worse if blocks computed concurrently**
    - Leads to one sample delay *per block*
- **Block-by-block computation requires additional algorithm to achieve minimum delay ordering**

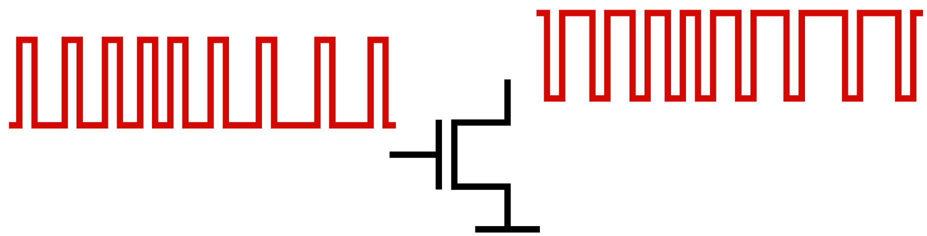
CppSim provides automatic minimum delay ordering and allows user specified ordering

Time-Based Circuits

- **Traditional analog circuits utilize voltage and current with bandwidth constrained signaling**



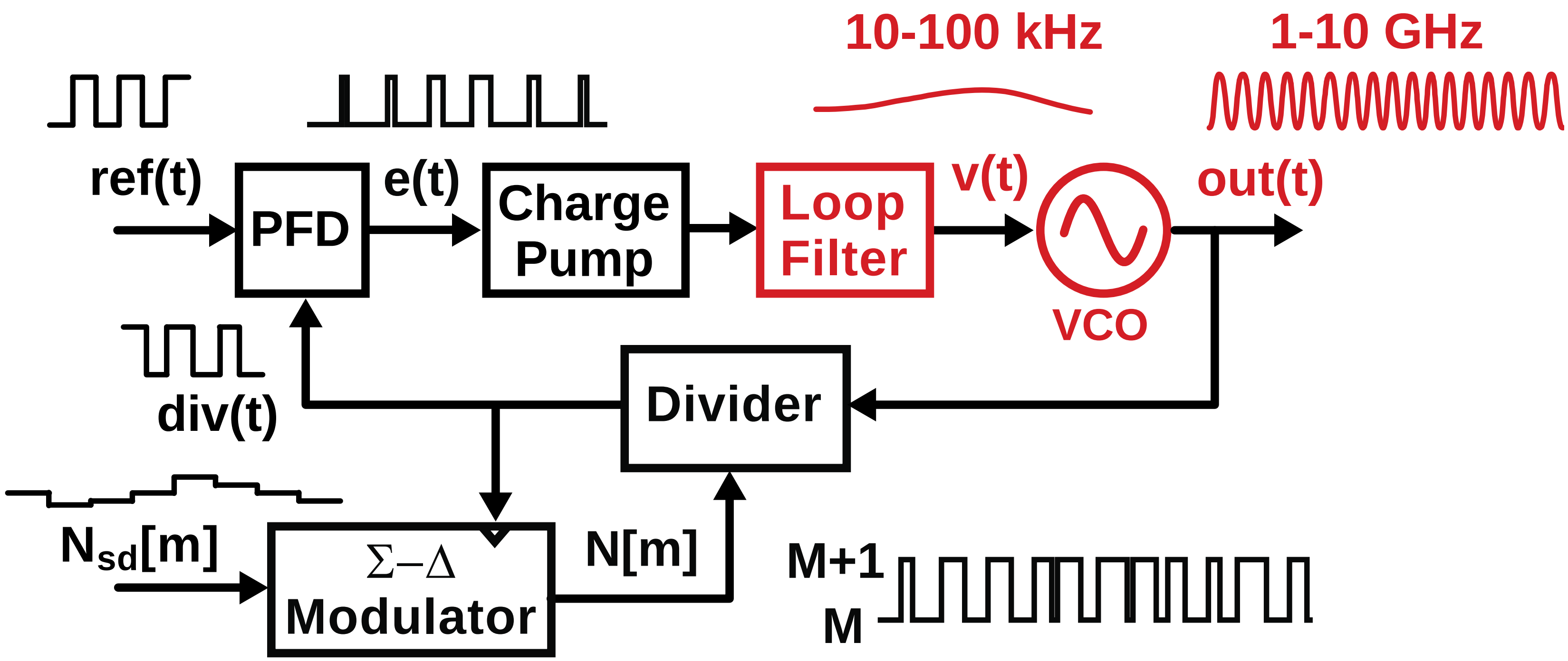
- **Time-based circuits utilize the timing of edges produced by “digital” circuits**



  - **Modern CMOS processes are offering faster edge rates and lower delay through digital circuits**

High bandwidth of time-based circuits creates challenges for high speed simulation

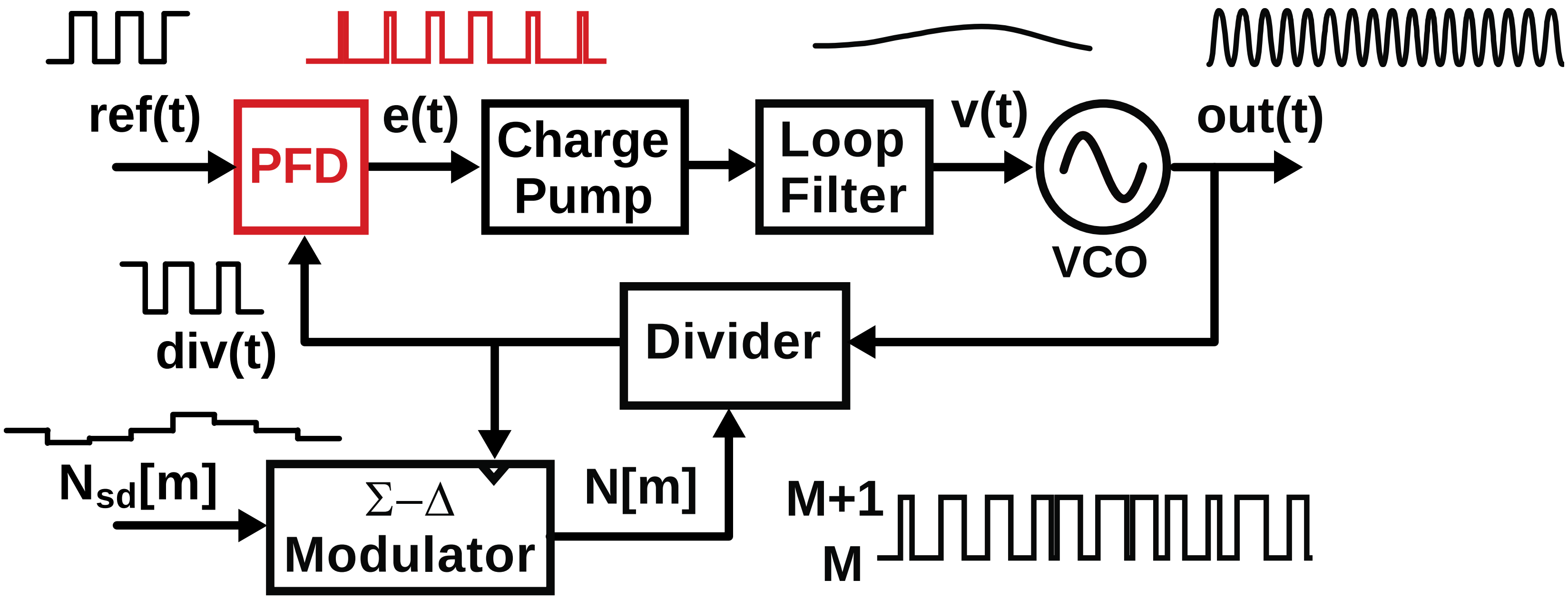
A Common Time-Based Circuit



- Consider a fractional-N synthesizer as a prototypical time-based circuit
  - High output frequency ➡ High sample rate
  - Long time constants ➡ Long time span for transients

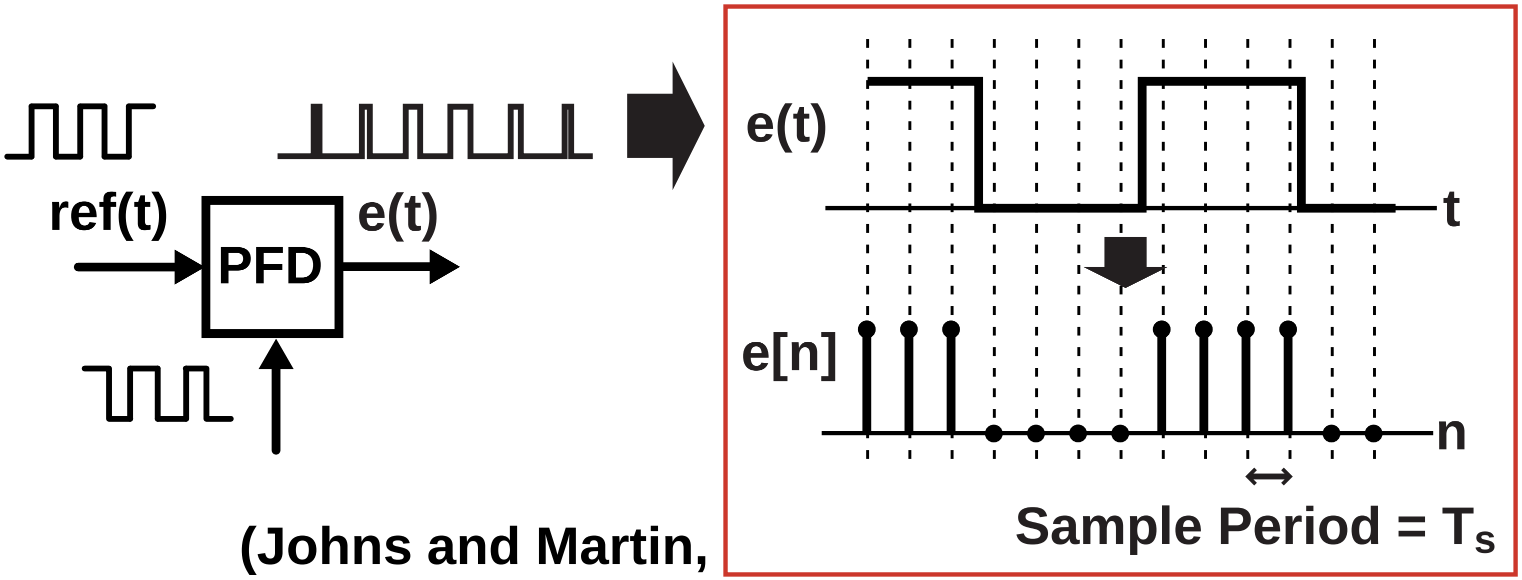
Large number of simulation time steps required

Continuously Varying Edges Lead to Accuracy Issues



- PFD output has very high bandwidth
  - Difficult to achieve high accuracy within a conventional discrete-time or SPICE level simulator
- Non-periodic dithering of divider complicates matters
  - Periodic, steady-state methods do not apply

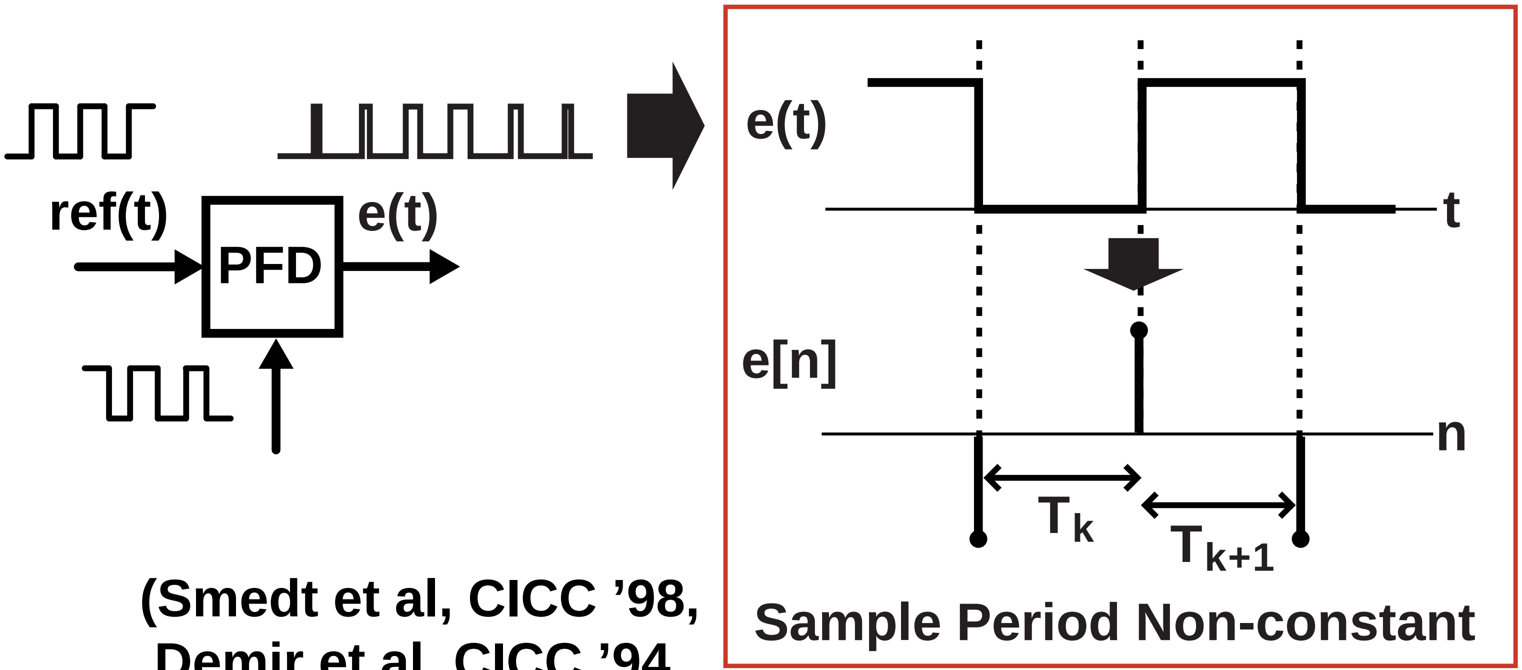
Consider A Classical Constant-Time Step Method



(Johns and Martin,
Analog Integrated Circuit Design)

- Directly sample the PFD output according to the simulation sample period
  - Simple, fast, readily implemented in Matlab, Verilog, C++
- Issue – quantization noise is introduced
  - This noise can overwhelm the PLL noise sources we are trying to simulate

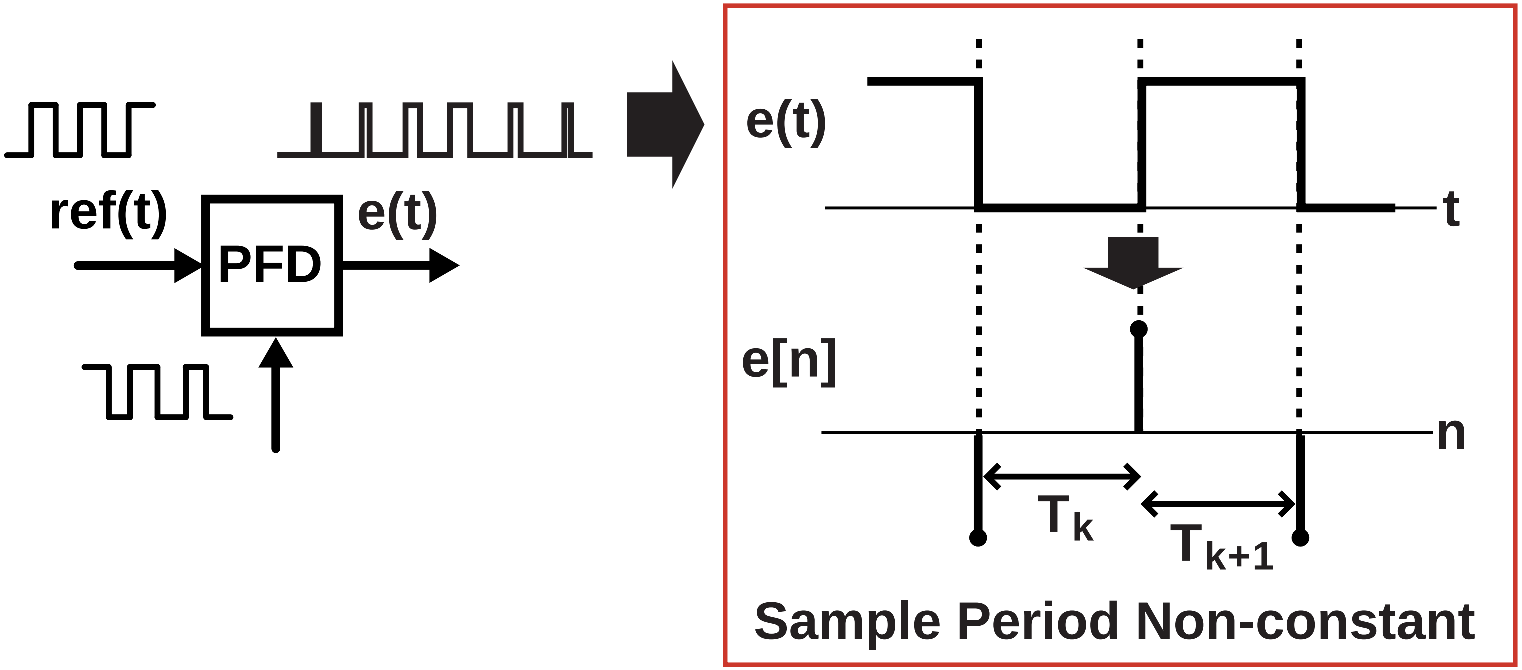
Alternative: Event Driven Simulation



(Smedt et al, CICC '98,
Demir et al, CICC '94,
Hinz et al, Circuits and Systems '00)

- **Set simulation time samples at PFD edges**
  - **Sample rate can be lowered to edge rate!**

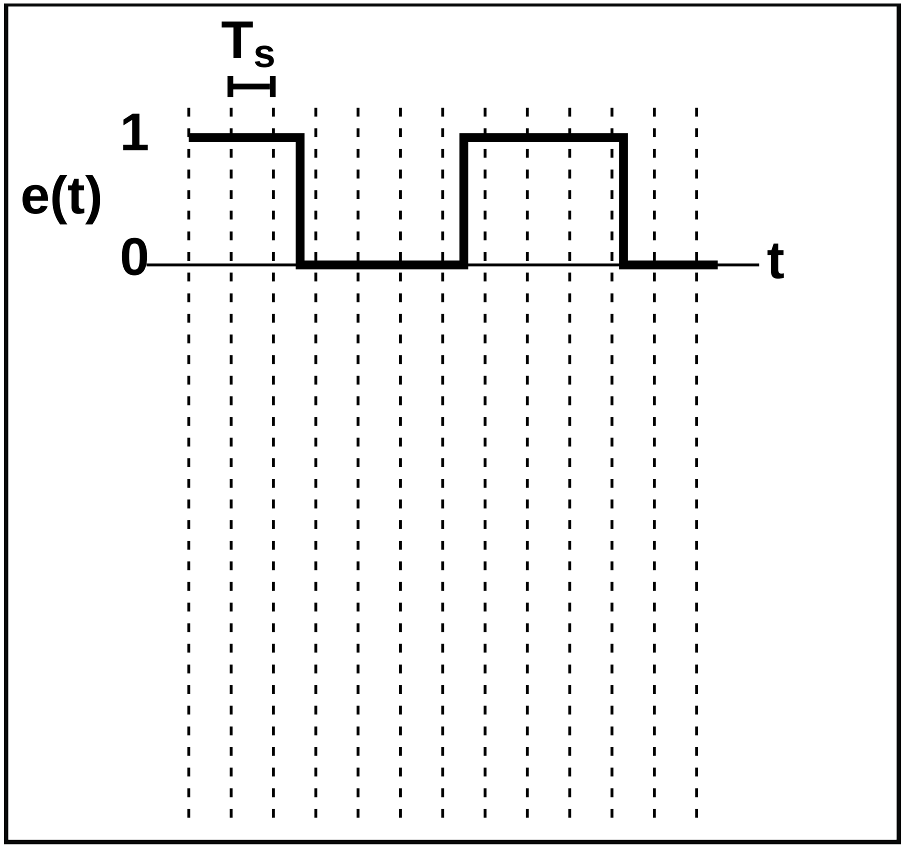
Issue: Non-Constant Time Step Brings Complications

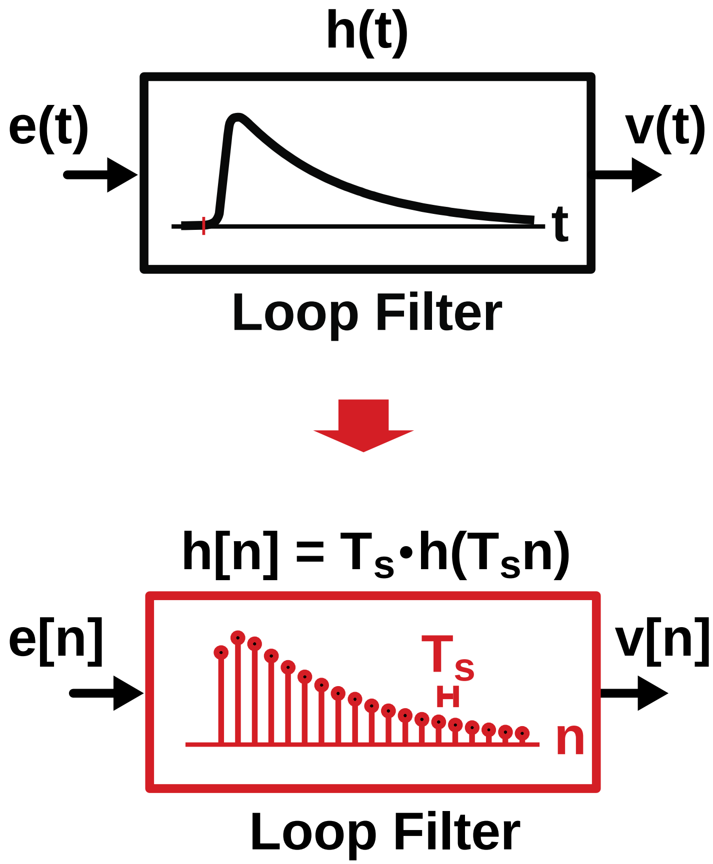


- Filters and noise sources must account for varying time step in their code implementations
- Spectra derived from mixing and other operations can display false simulation artifacts
- Setting of time step becomes progressively complicated if multiple time-based circuits simulated at once

Is there a better way?

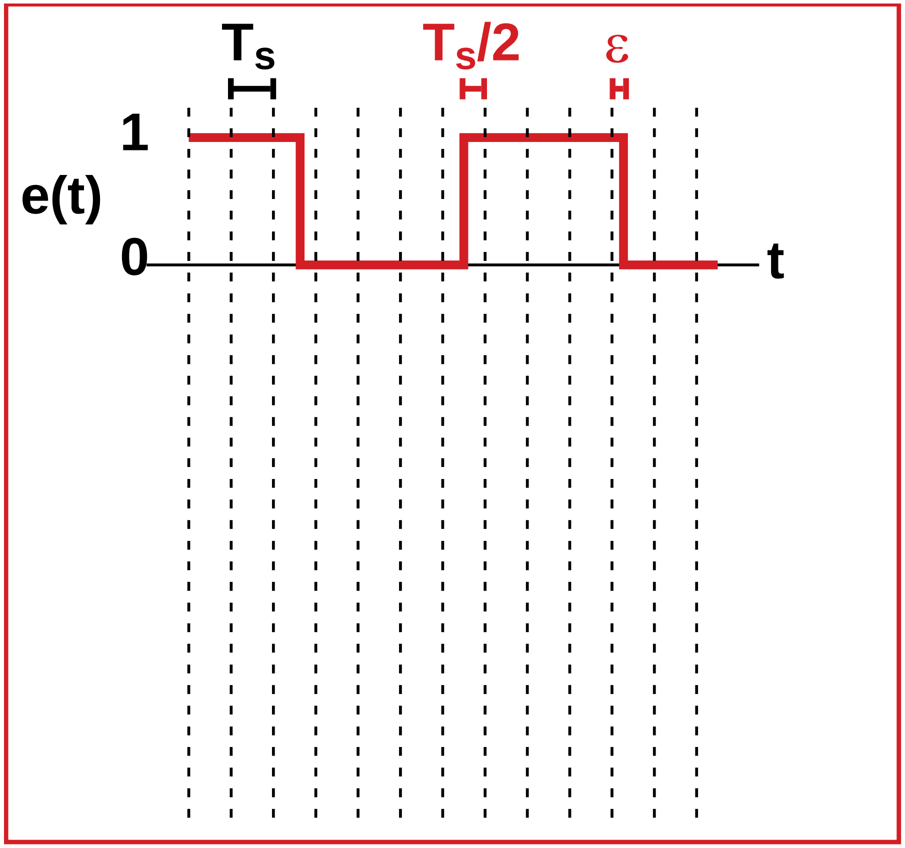
Proposed Approach: Use Constant Time Step

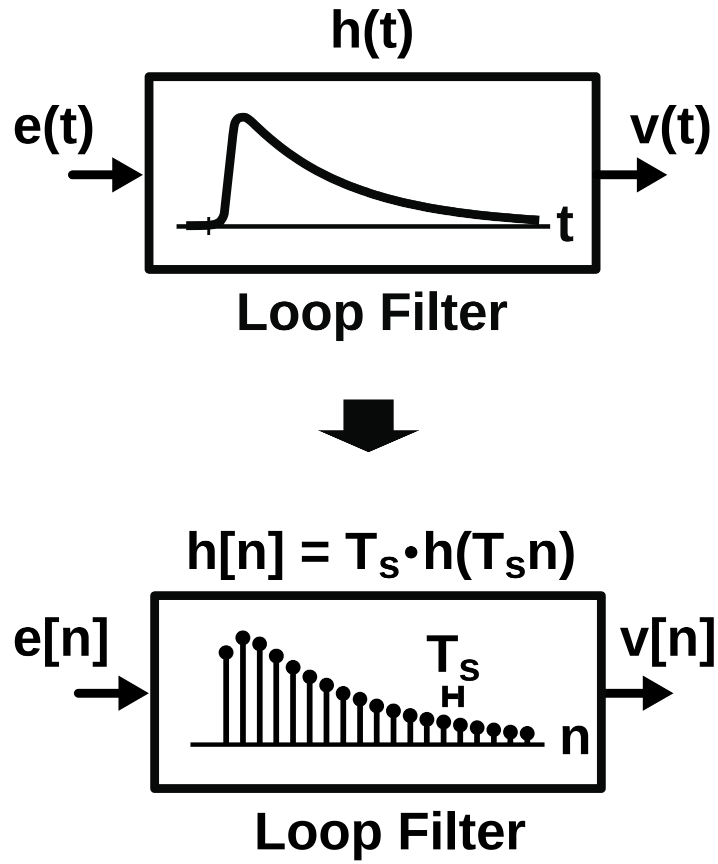




- Straightforward CT to DT transformation of filter blocks
  - Use bilinear transform or impulse invariance methods
- Overall computation framework is fast and simple
  - Simulator can be based on Verilog, Matlab, C++

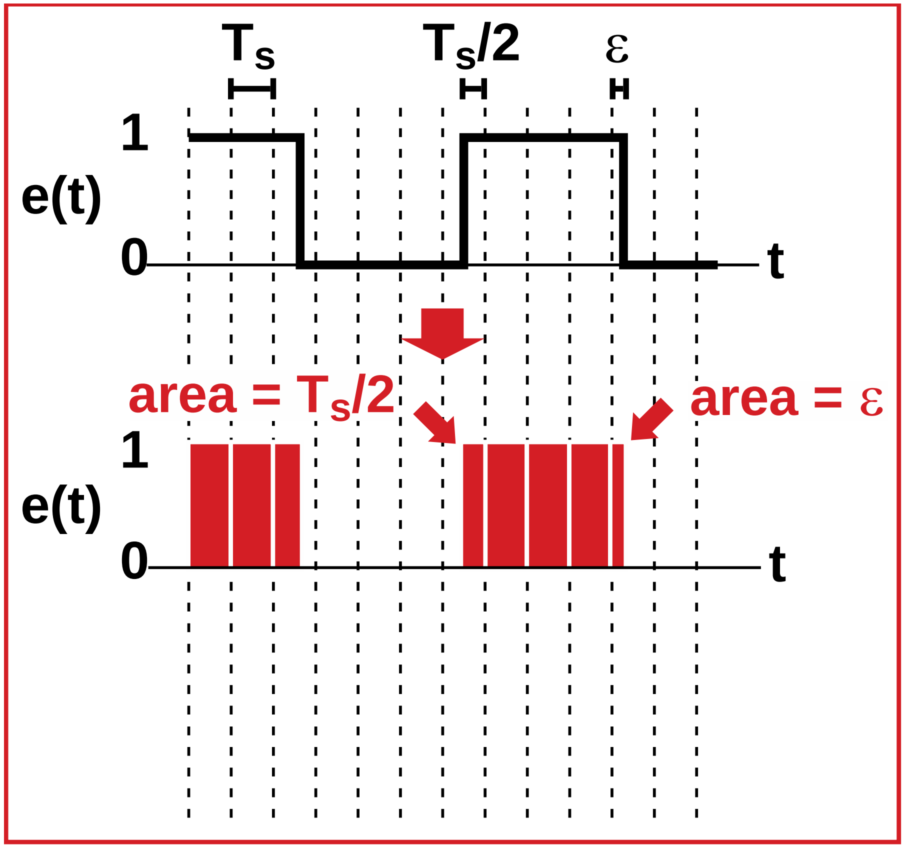
Problem: Quantization Noise at PFD Output

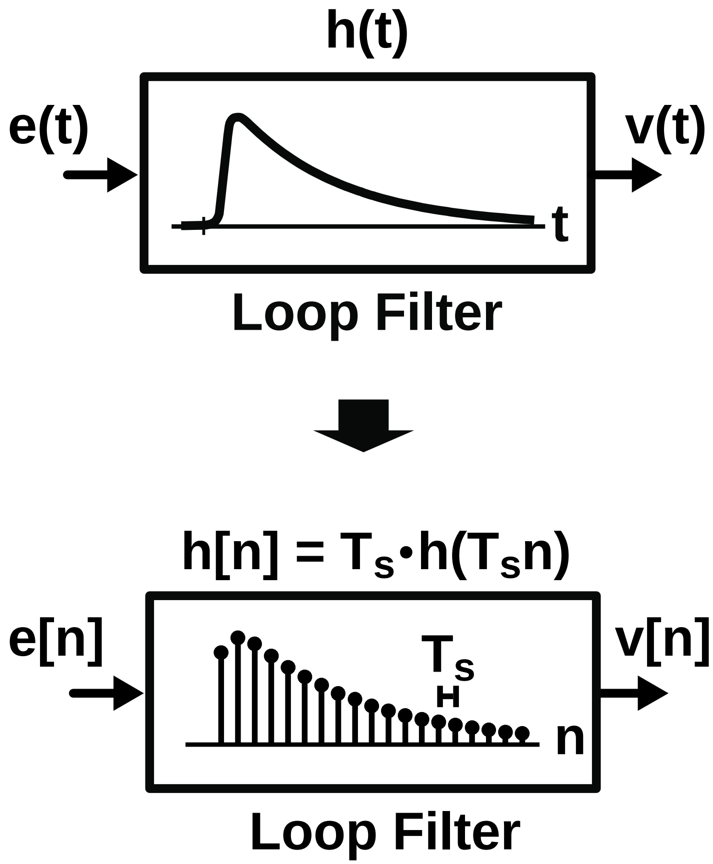




- **Edge locations of PFD output are quantized**
  - **Resolution set by time step: T_S**
- **Reduction of T_S leads to long simulation times**

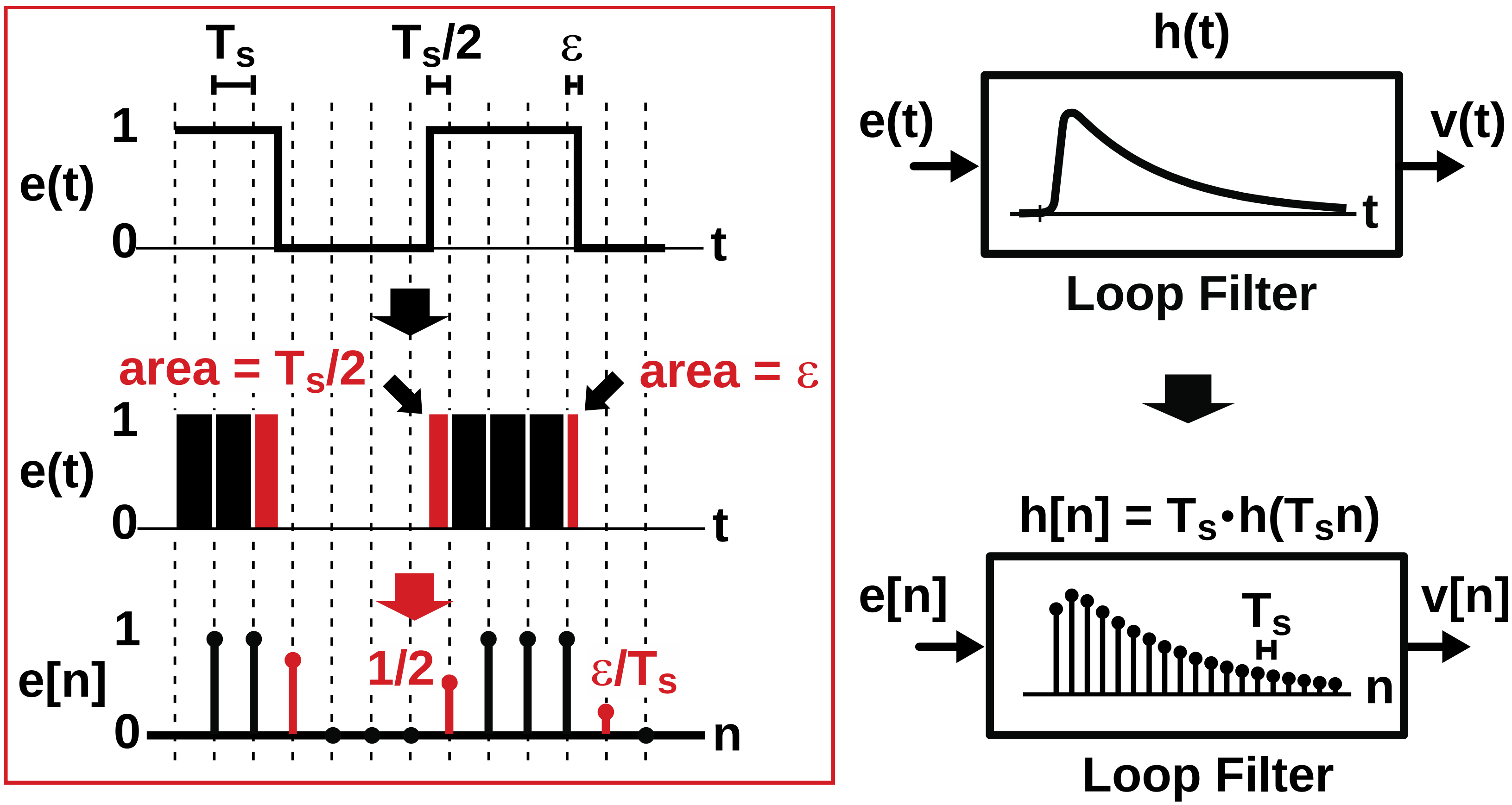
Proposed Approach: View as Series of Pulses





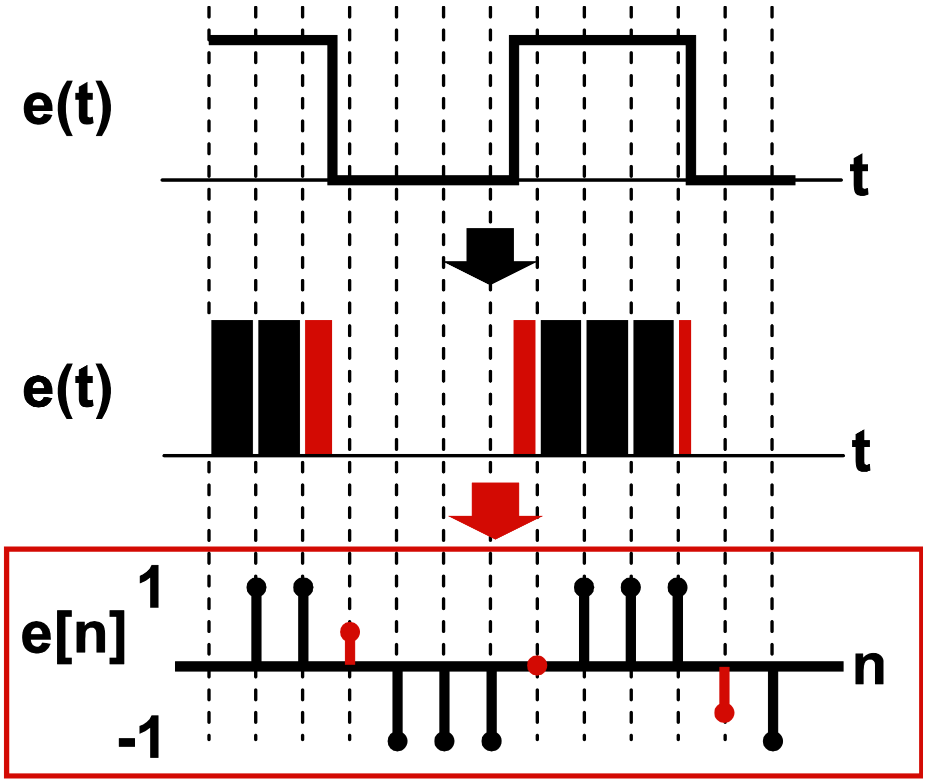
- Area of each pulse set by edge locations
- Key observations:
  - Pulses look like impulses to loop filter
  - Impulses are parameterized by their area and time offset

Proposed Area Conservation Method



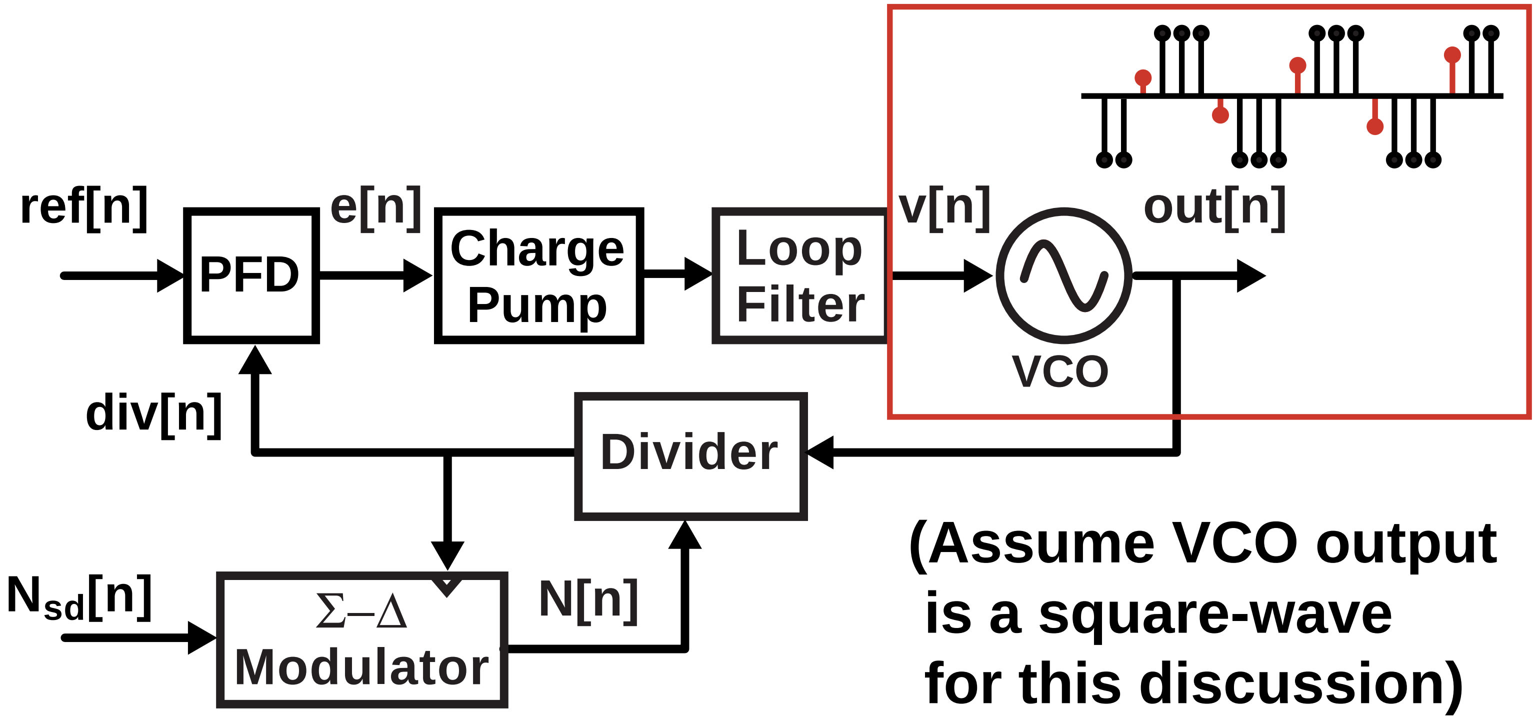
- Set e[n] samples according to pulse areas
  - Leads to very accurate results
  - Fast computation

Double_Interp Protocol



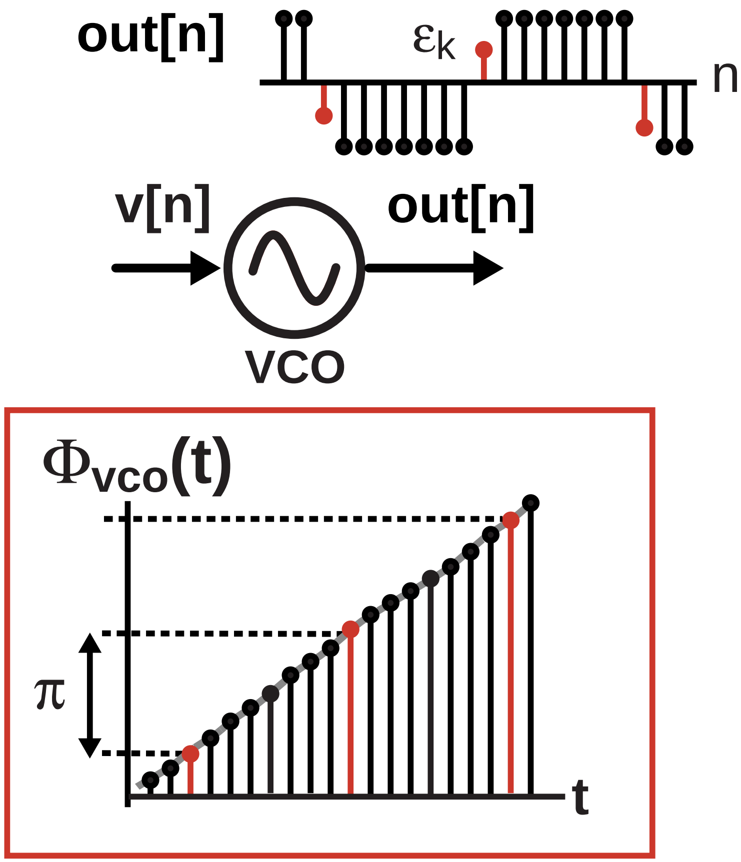
- Protocol sets signal samples to -1 or 1 except for transitions
  - Transition values between -1 and 1 are directly related to the edge time location
  - Can be implemented in C++, Verilog, and Matlab/Simulink

VCO is a Key Block for Double_Interp Encoding



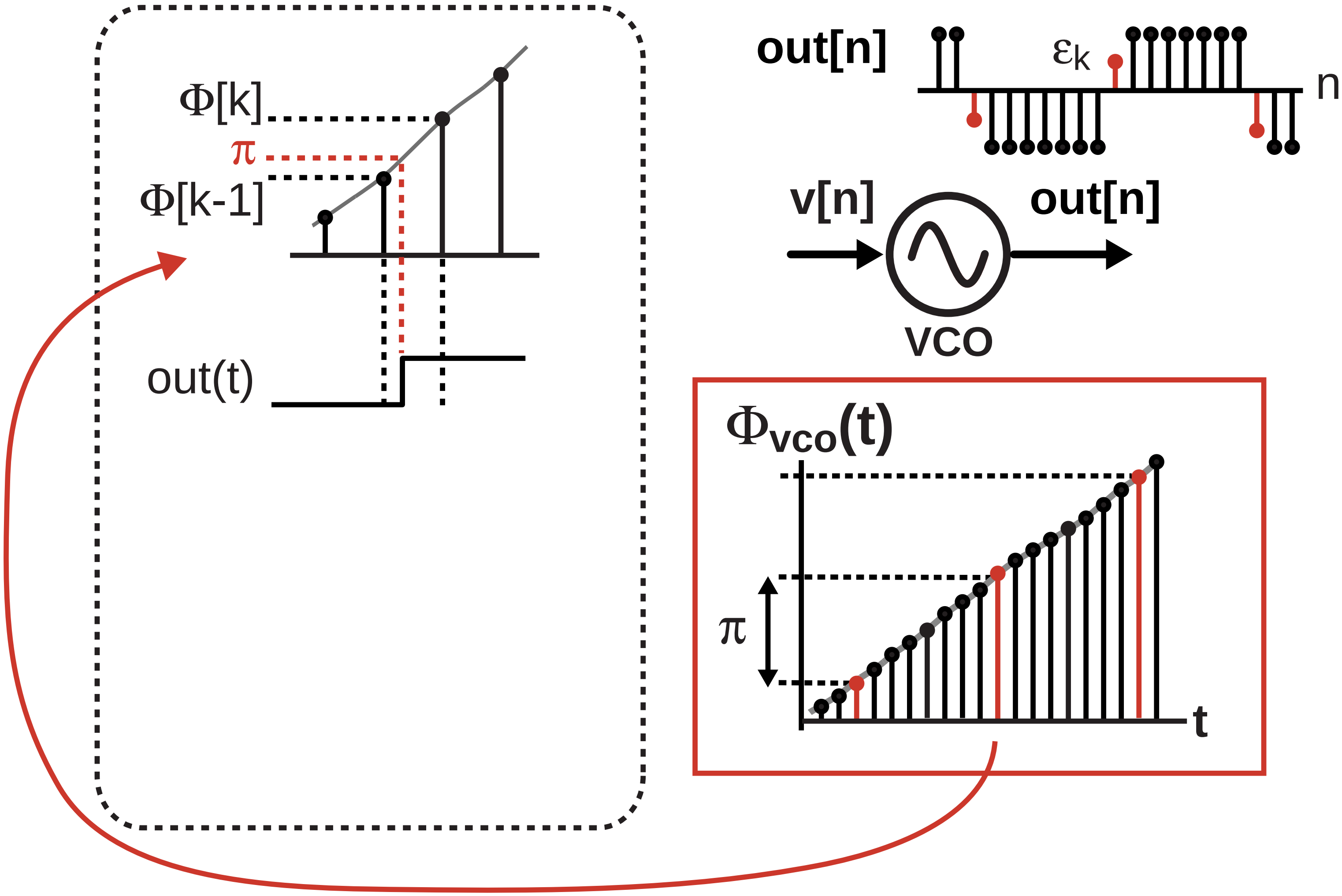
- The VCO block is the key translator from a bandlimited analog input to an edge-based waveform
  - We can create routines in the VCO that calculate the edge times of the output and encode their values using the double_interp protocol

Calculation of Transition Time Values



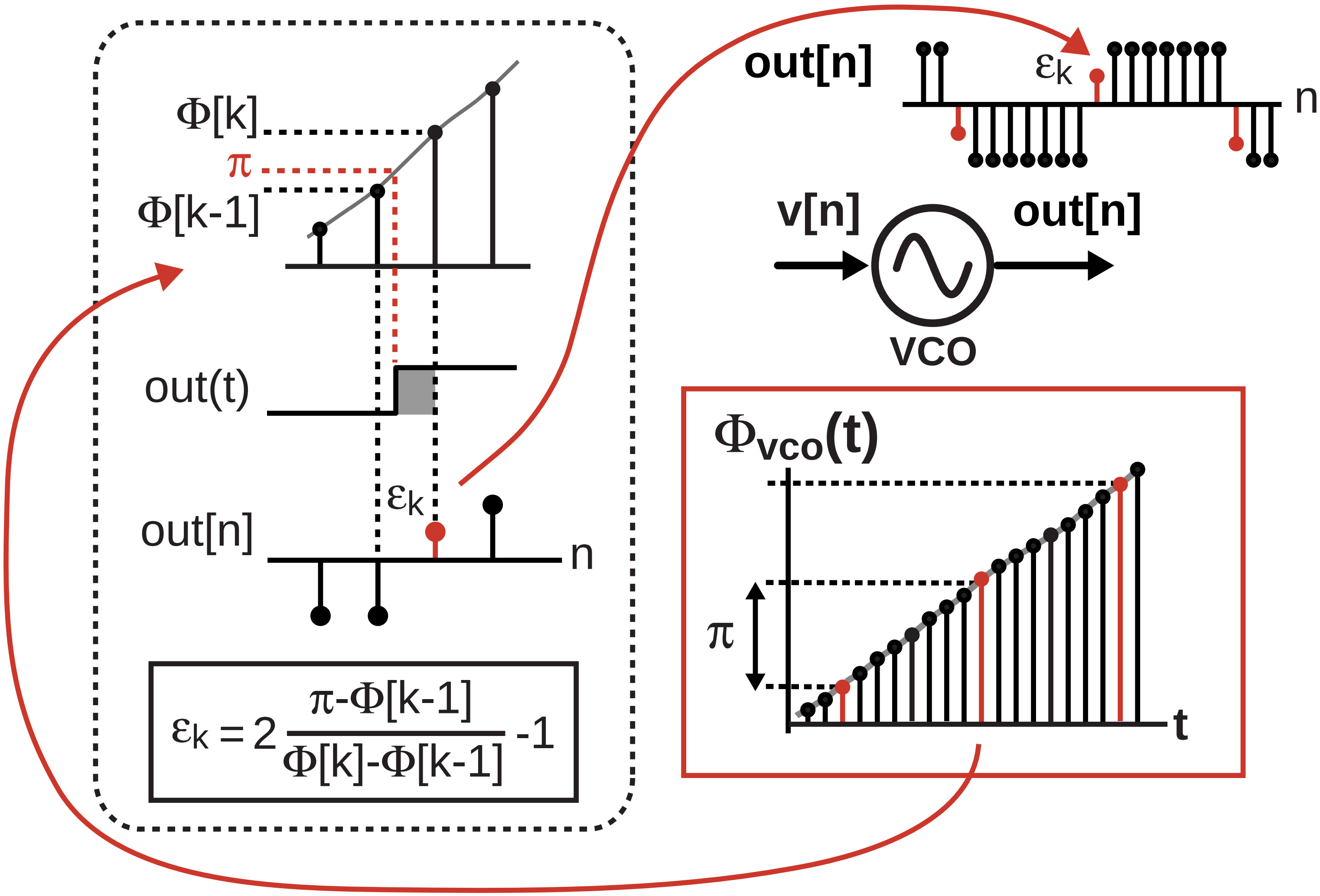
- **Model VCO based on its phase**

Calculation of Transition Time Values (cont.)



- **Determine output transition time according to phase**

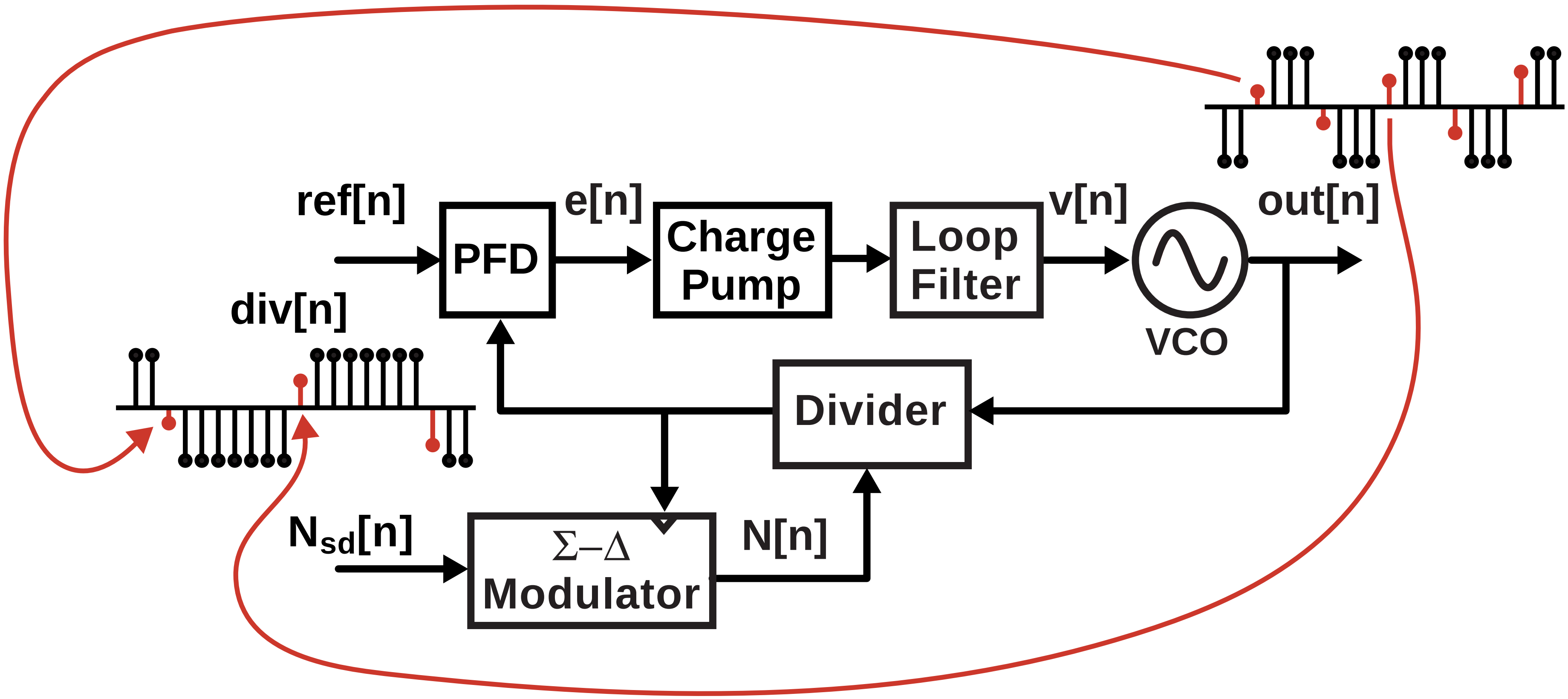
Calculation of Transition Time Values (cont.)



$$\varepsilon_k = 2\,\frac{\pi - \Phi[k-1]}{\Phi[k]-\Phi[k-1]} - 1$$

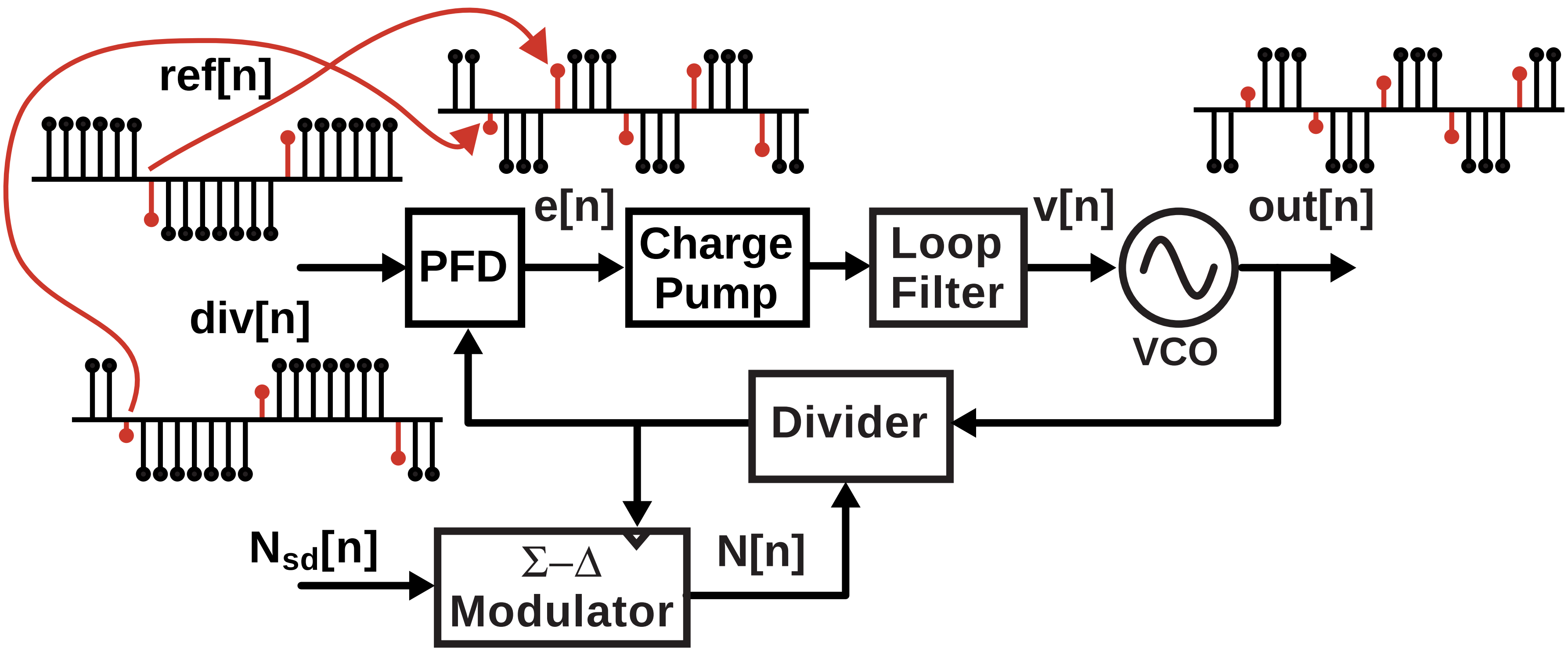
- **Use first order interpolation to determine transition value**

Processing of Edges using Double_Interp Protocol



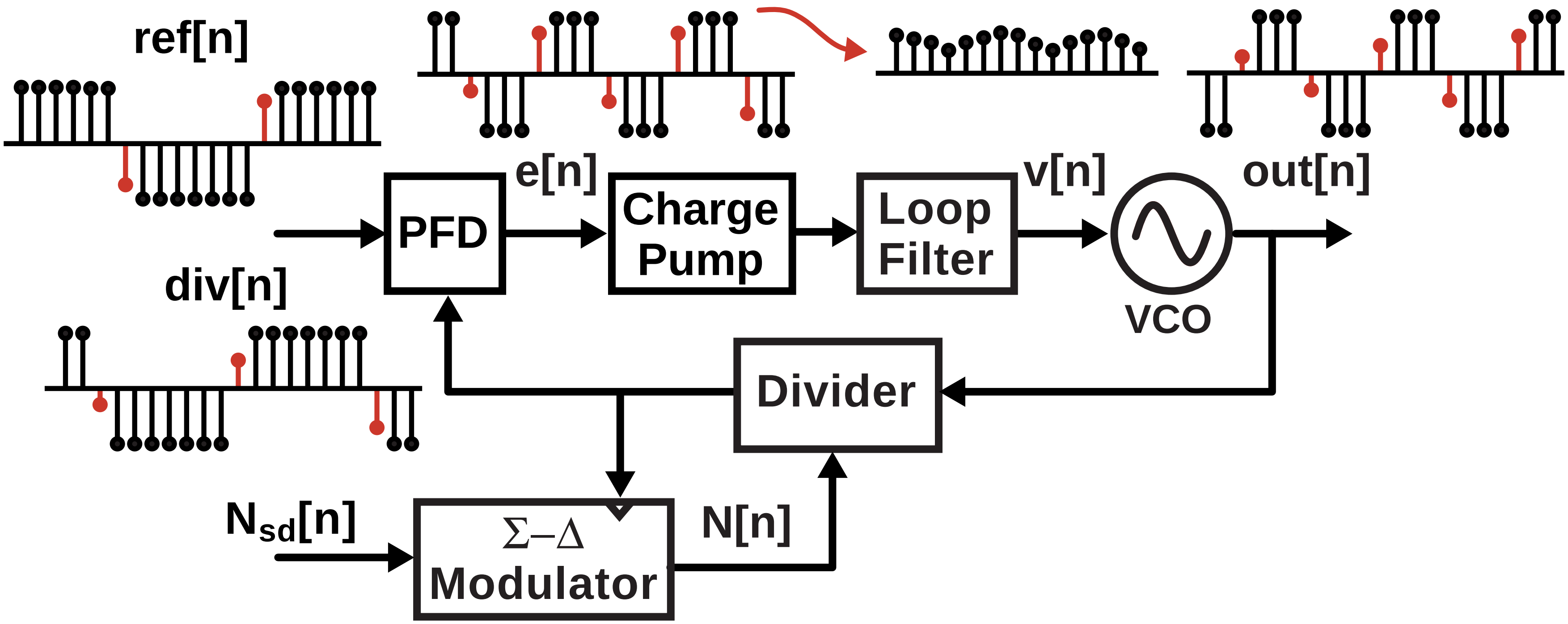
- Frequency divider block simply passes a sub-sampling of edges based on the VCO output and divide value

Processing of Edges using Double_Interp Protocol



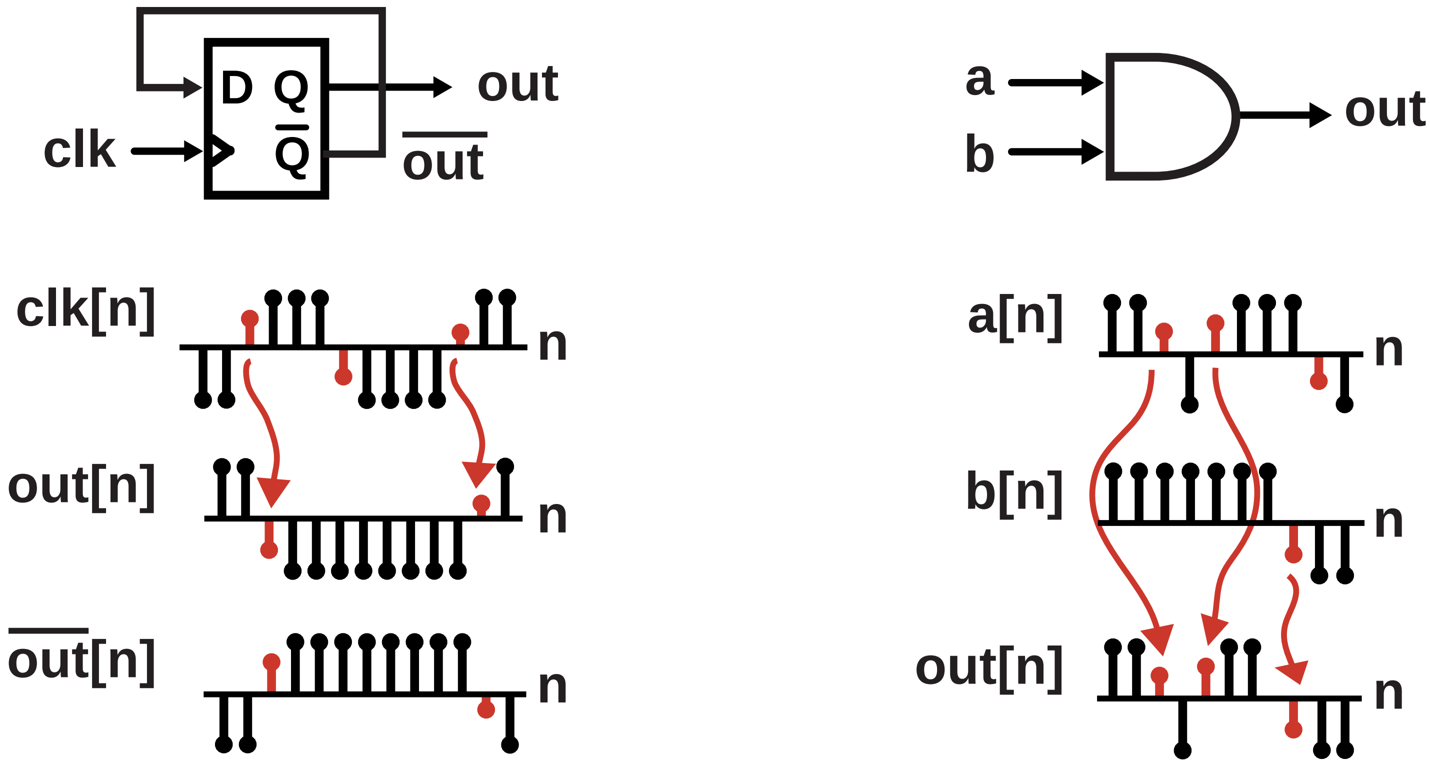
- **Phase Detector compares edges times between reference and divided output and then outputs pulses that preserve the time differences**

Processing of Edges using Double_Interp Protocol



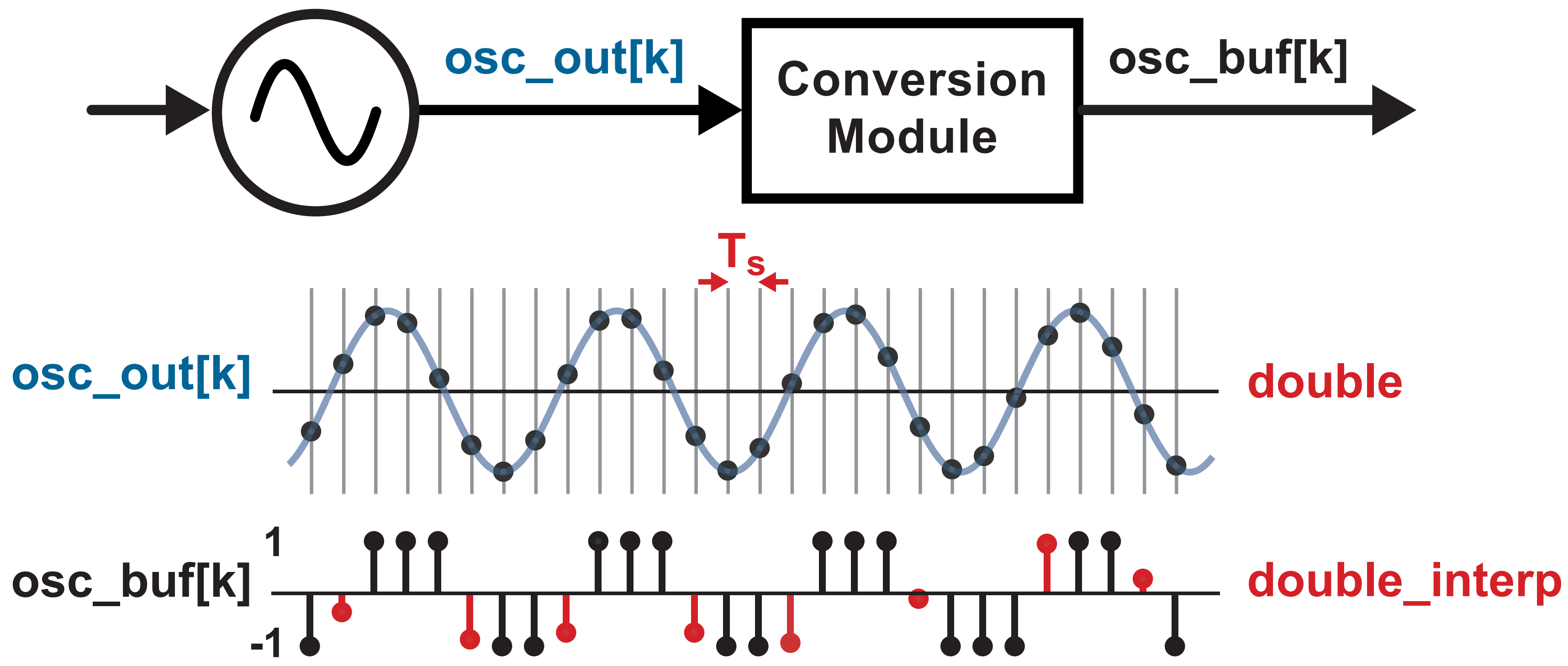
- **Charge Pump and Loop filter operation is straightforward to model**
  - **Simply filter pulses from phase detector as discussed earlier**

Using the Double_Interp Protocol with Digital Gates



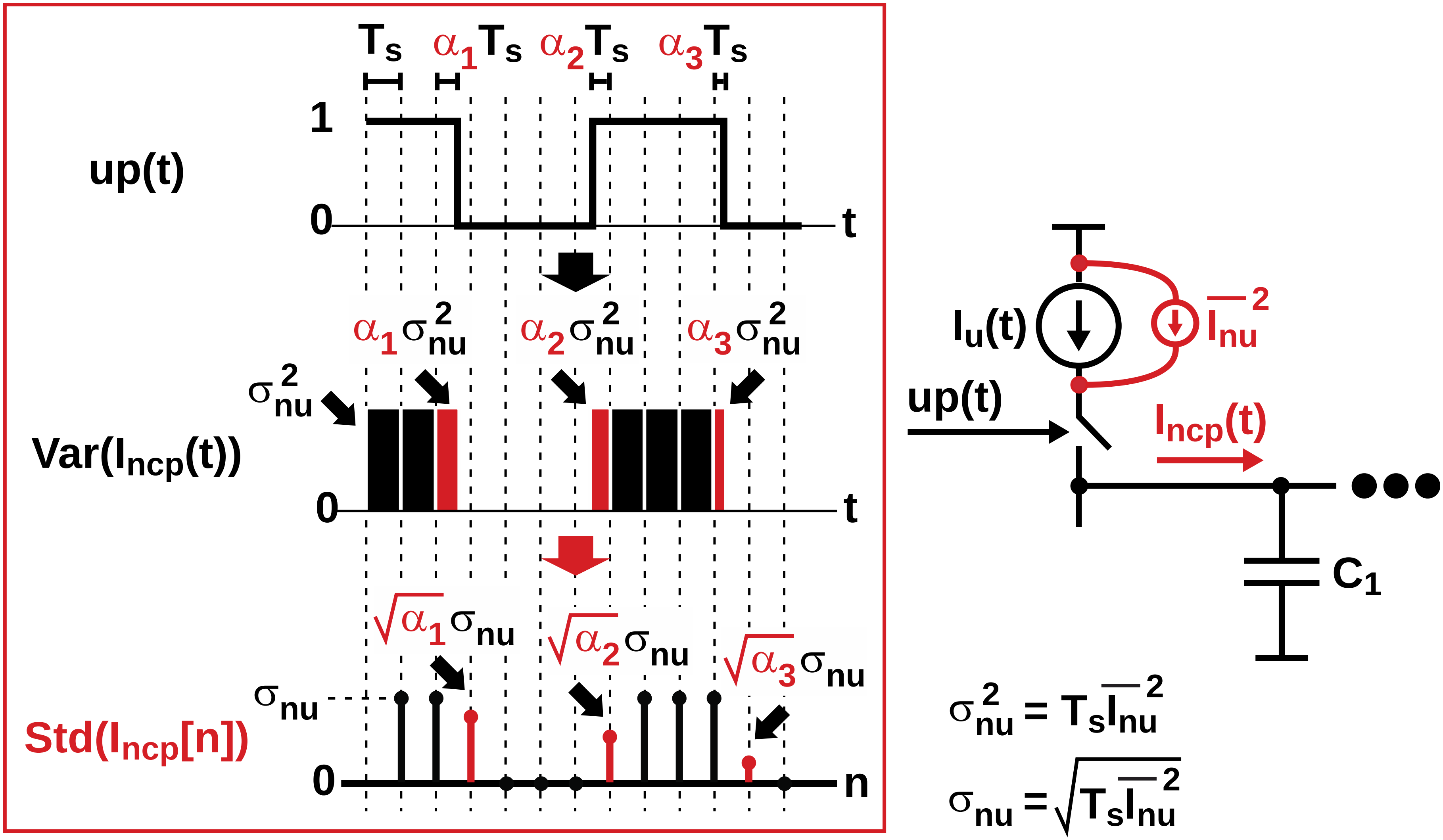
- **Relevant timing information contained in the input that causes the output to transition**
  - **Determine which input causes the transition, then pass its transition value to the output**

Using the Double_Interp Protocol with Sine Waves



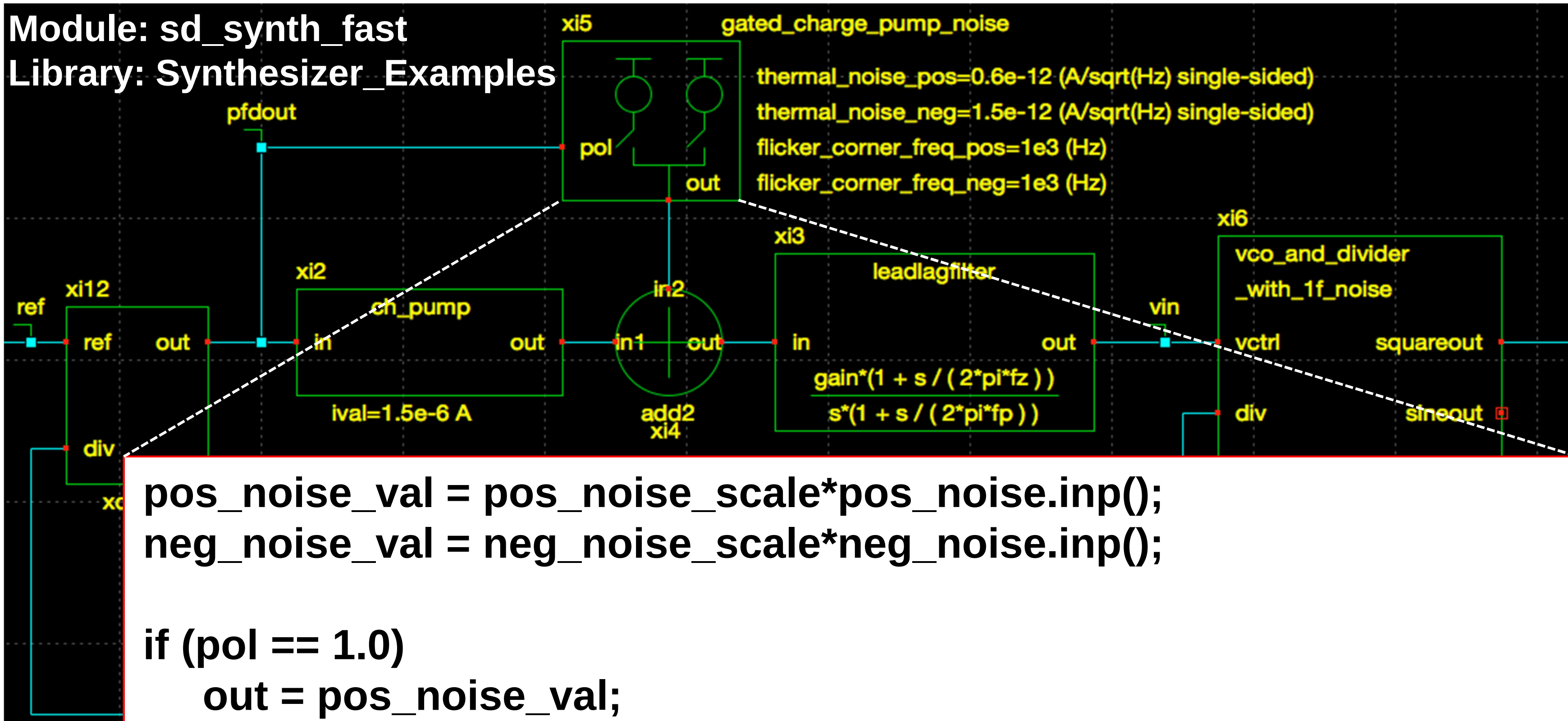
- **In some systems we must deal directly with sine waves**
  - **An explicit conversion module should be utilized**
    - We can convert to double_interp protocol using a similar interpolation technique as described earlier
  - **See gmsk_limitamp module within GMSK_Example library**
    - Used in module gmsk_pll_transmitter in the same library

Using the Double_Interp Protocol with Noise



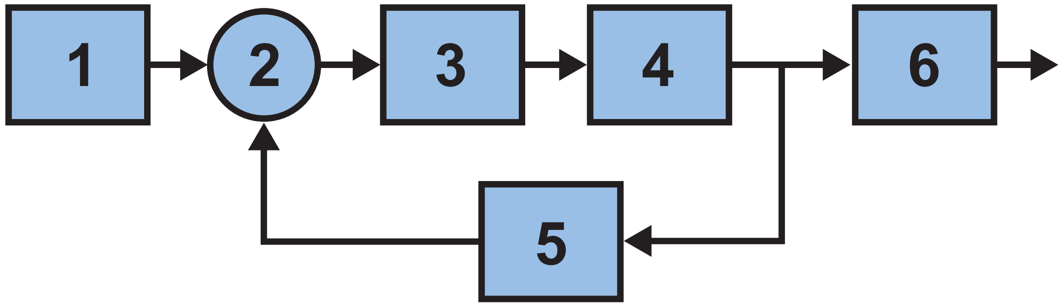
- **Standard deviation of noise samples impacted by edges**
  - **Standard deviation scaled by sqrt of "α" value for edge time**
    - "α" value determined using double_interp protocol value

Example: Charge Pump Noise for XOR PD



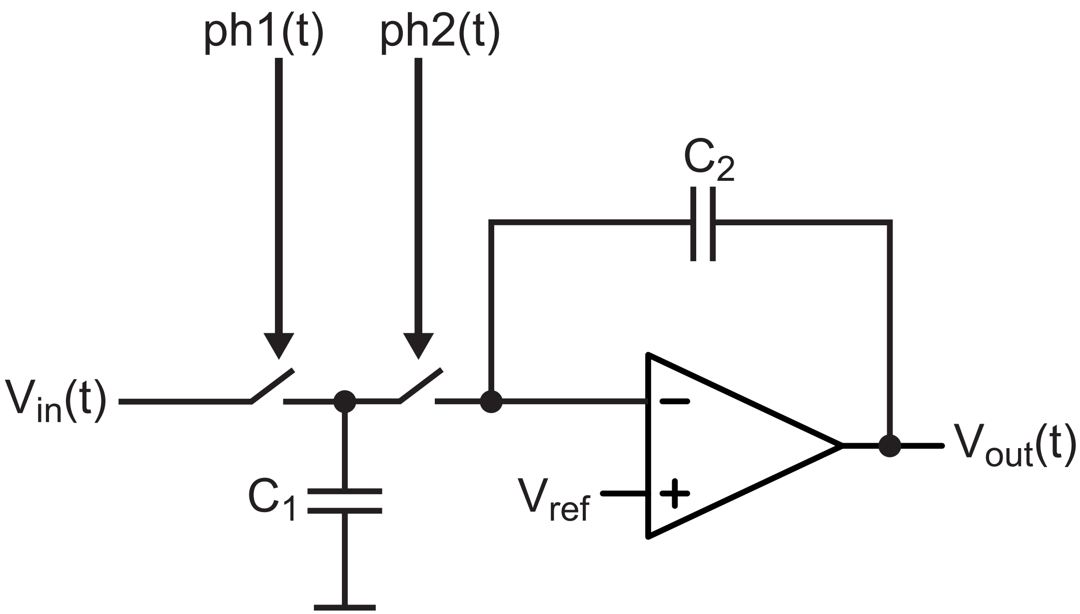

```
pos_noise_val = pos_noise_scale*pos_noise.inp();
neg_noise_val = neg_noise_scale*neg_noise.inp();

if (pol == 1.0)
    out = pos_noise_val;
else if (pol == -1.0)
    out = neg_noise_val;
else if (pol >= -1.0 && pol <= 1.0)
    {
    interp_val = (1.0 + pol)/2.0;
    out = sqrt(interp_val)*pos_noise_val + sqrt(1.0 - interp_val)*neg_noise_val;
    }
else
    out = 0.0;
```

Summary of Block-by-Block Computation Method



- Requires unilateral flow through blocks
- Impacts phase margin of feedback loops
  - Need $1/T_s$ >> bandwidth of feedback loop
  - Need proper ordering of blocks (automatic in CppSim)
- Constant time step simplifies simulation
  - Easier block descriptions
  - Frequency domain analysis become straightforward
  - Time-based signals handled with double_interp protocol

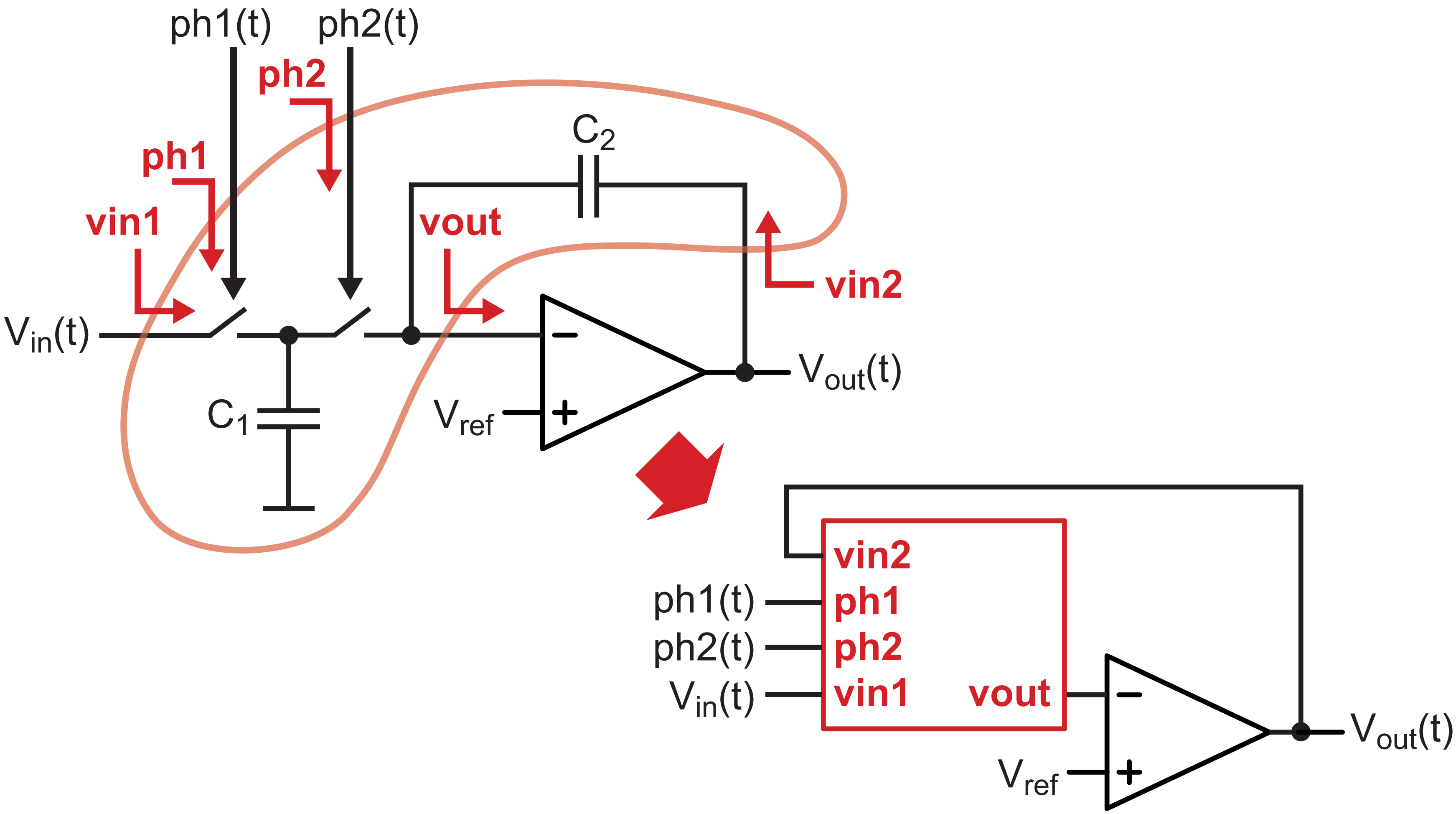
Simulation of Switched Capacitor Circuits



- **Capacitor network with switches can be modeled with unilateral flow blocks, but many practical issues:**
  - **Very challenging for beginners, tedious for experts**
  - **Difficult to check correctness of model**
  - **Difficult to investigate alternative architectures**

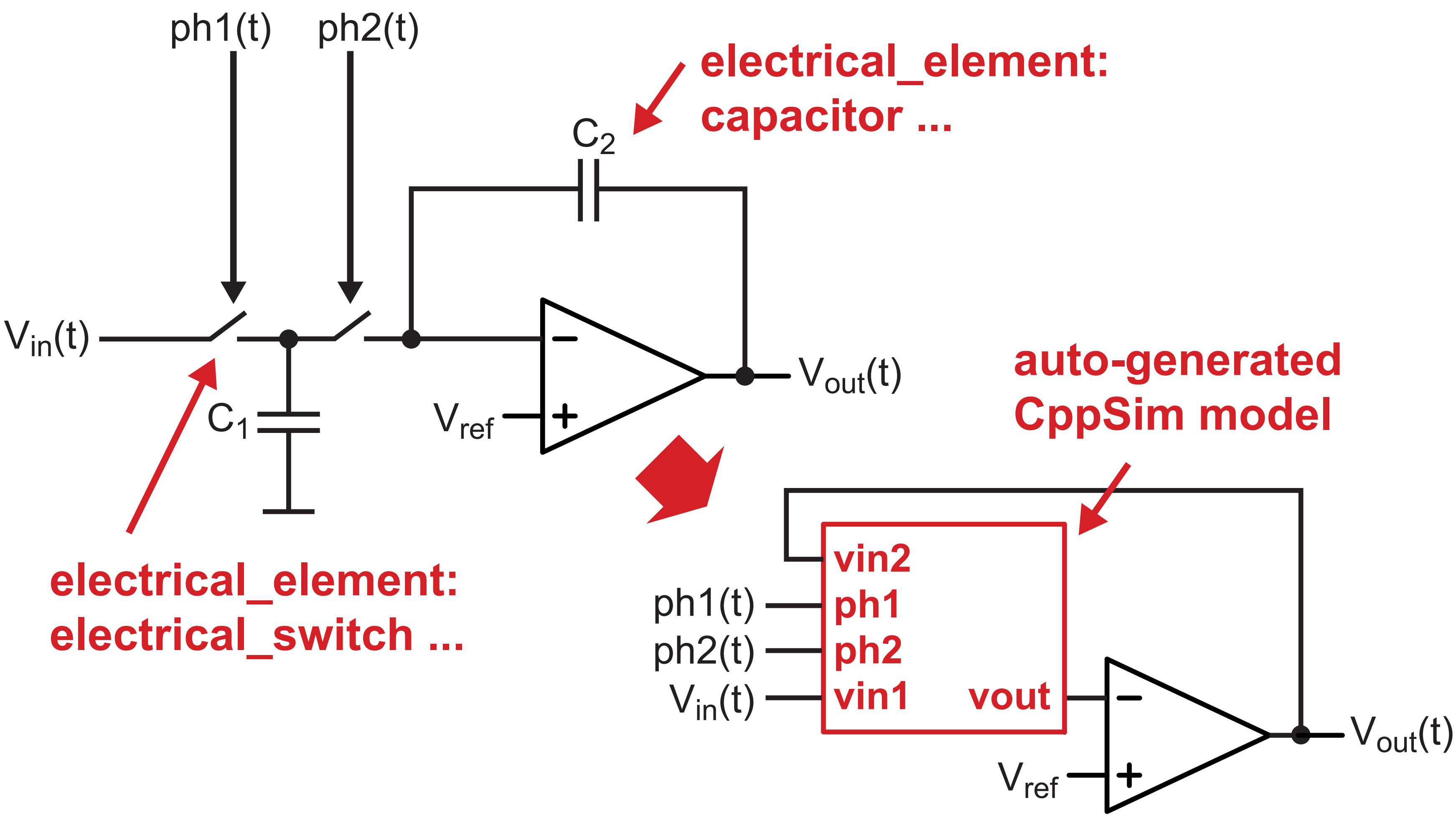
We need a way to automate the modeling process…

Automatic Unilateral Model Generation



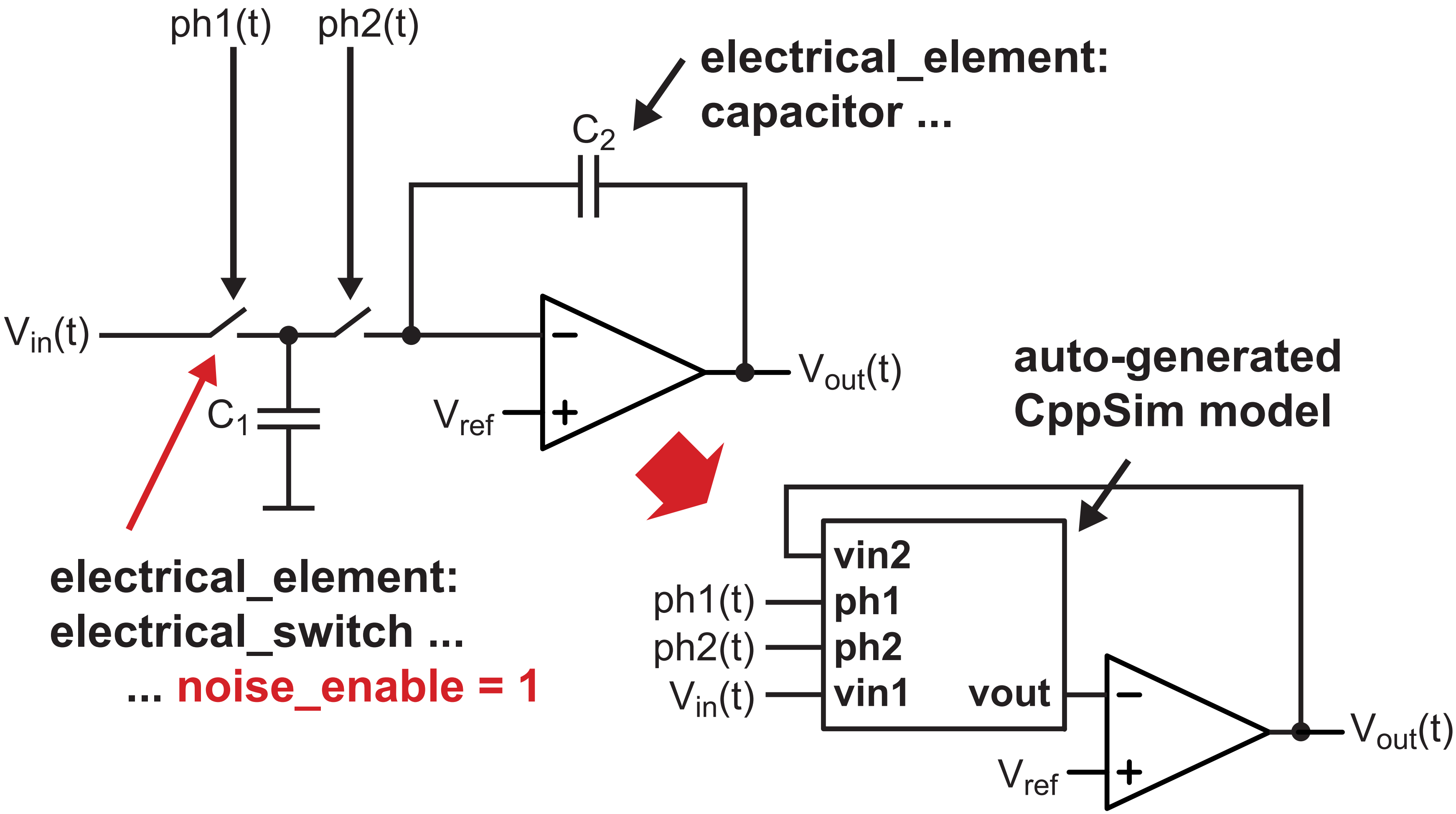
- **A linear network with switches can be represented as a state-space model with switch dependent matrices**
  - **An equivalent unilateral flow block is created**

CppSim Approach to Linear Networks with Switches



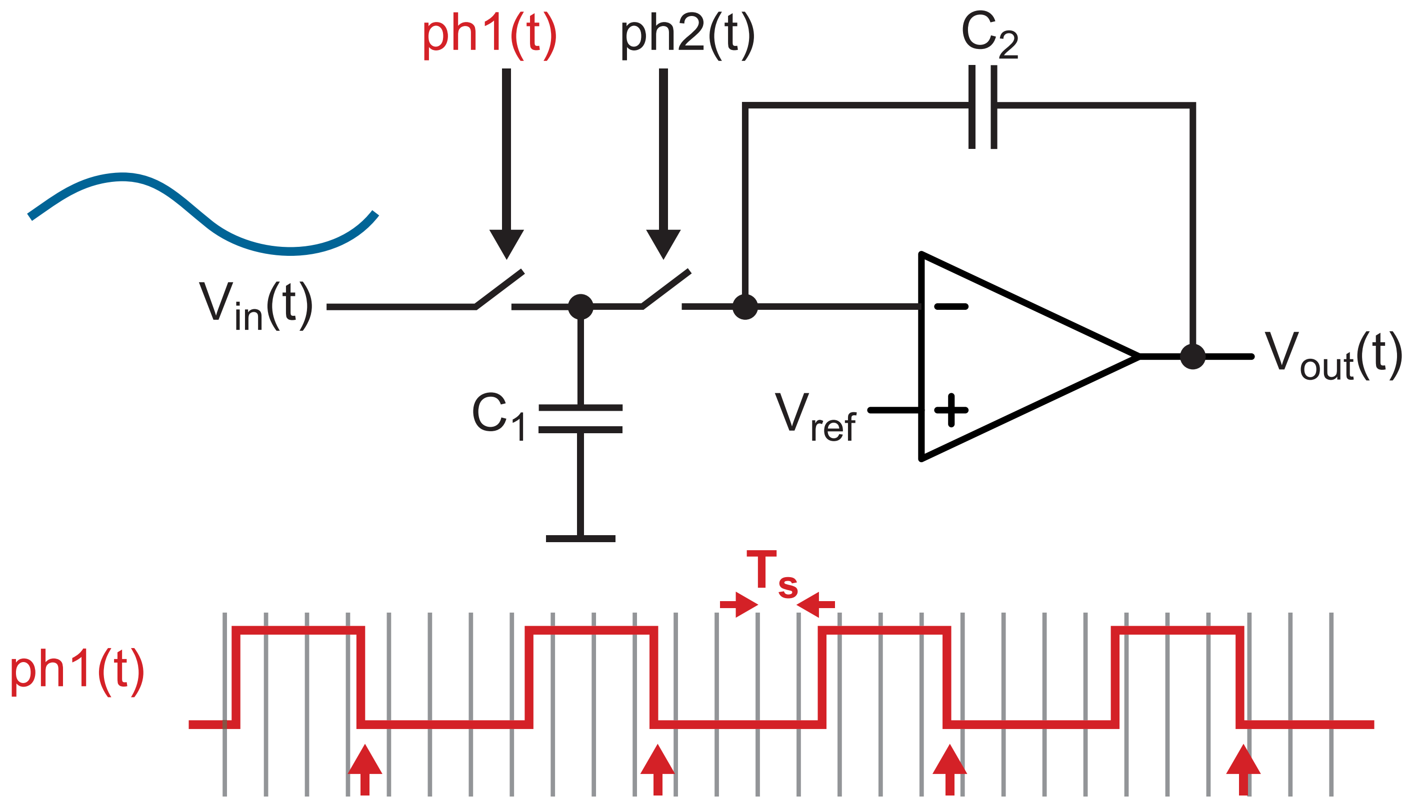
- **User specifies the CppSim model for linear elements, switches, and diodes using electrical_element: command**
  - **Draw the schematic and CppSim takes care of the rest!**

Transient Noise Analysis is Supported



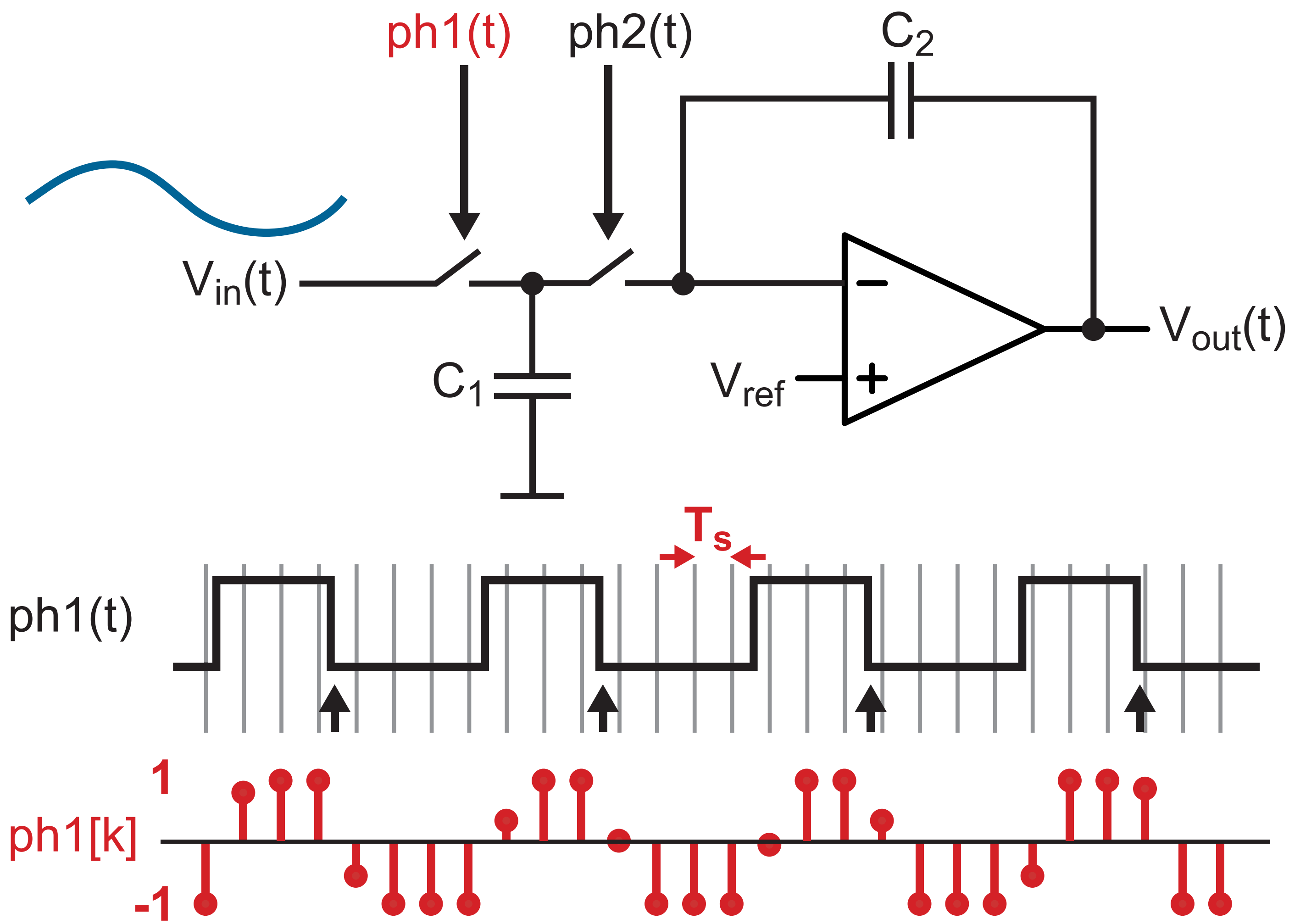
- Resistors, switches, voltage/current thermal + 1/f noise
- For kT/C noise, need adequately small time step, T_s
  - Accuracy requires $1/T_s$ > 20*bandwidth of switch settling time

Time Based Signals with Electrical Elements



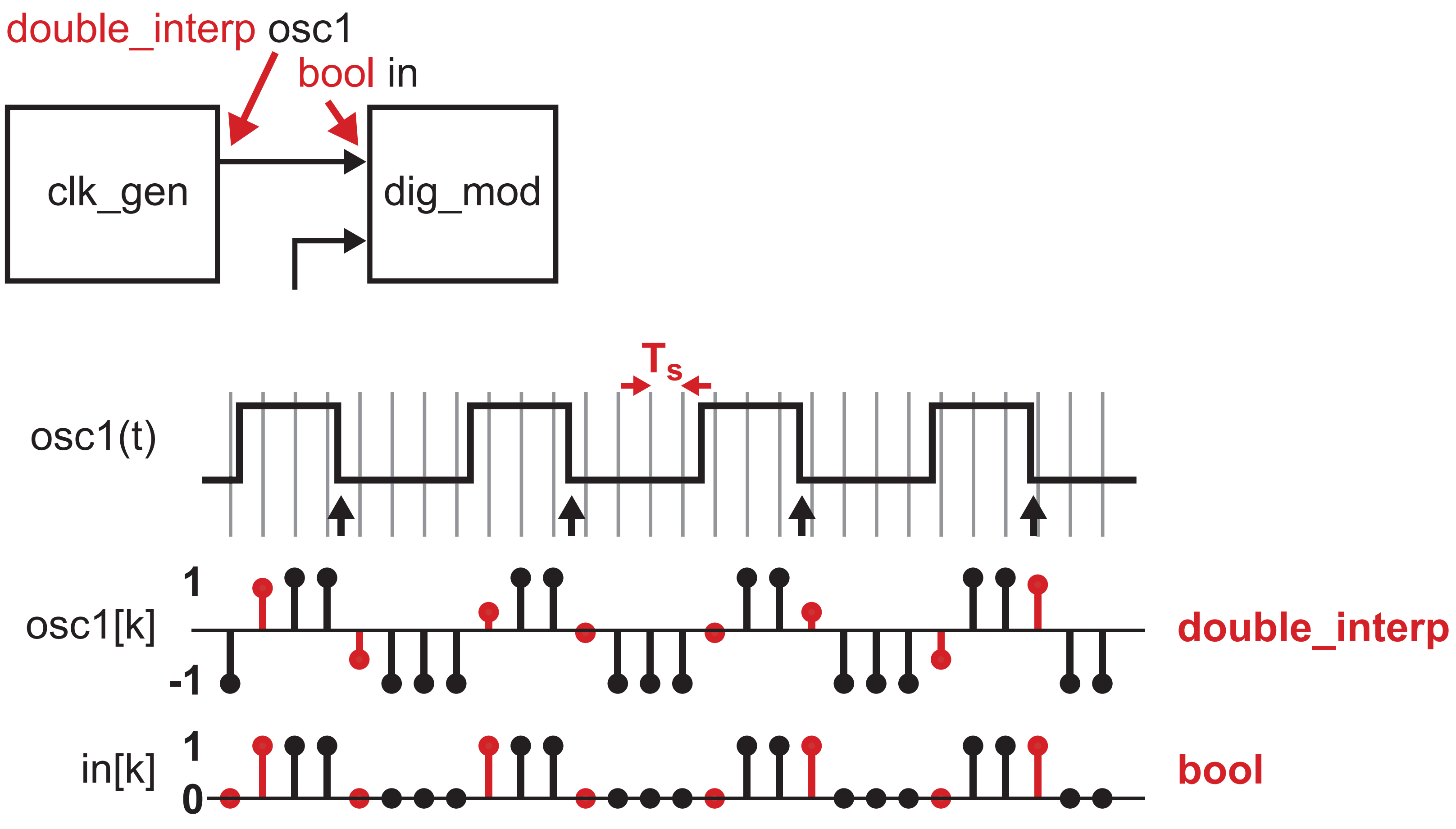
- Constant time step of CppSim could lead to quantization effects on sample times of clock edges
  - Would result in sampling errors of input waveform

Leverage Double_Interp Protocol



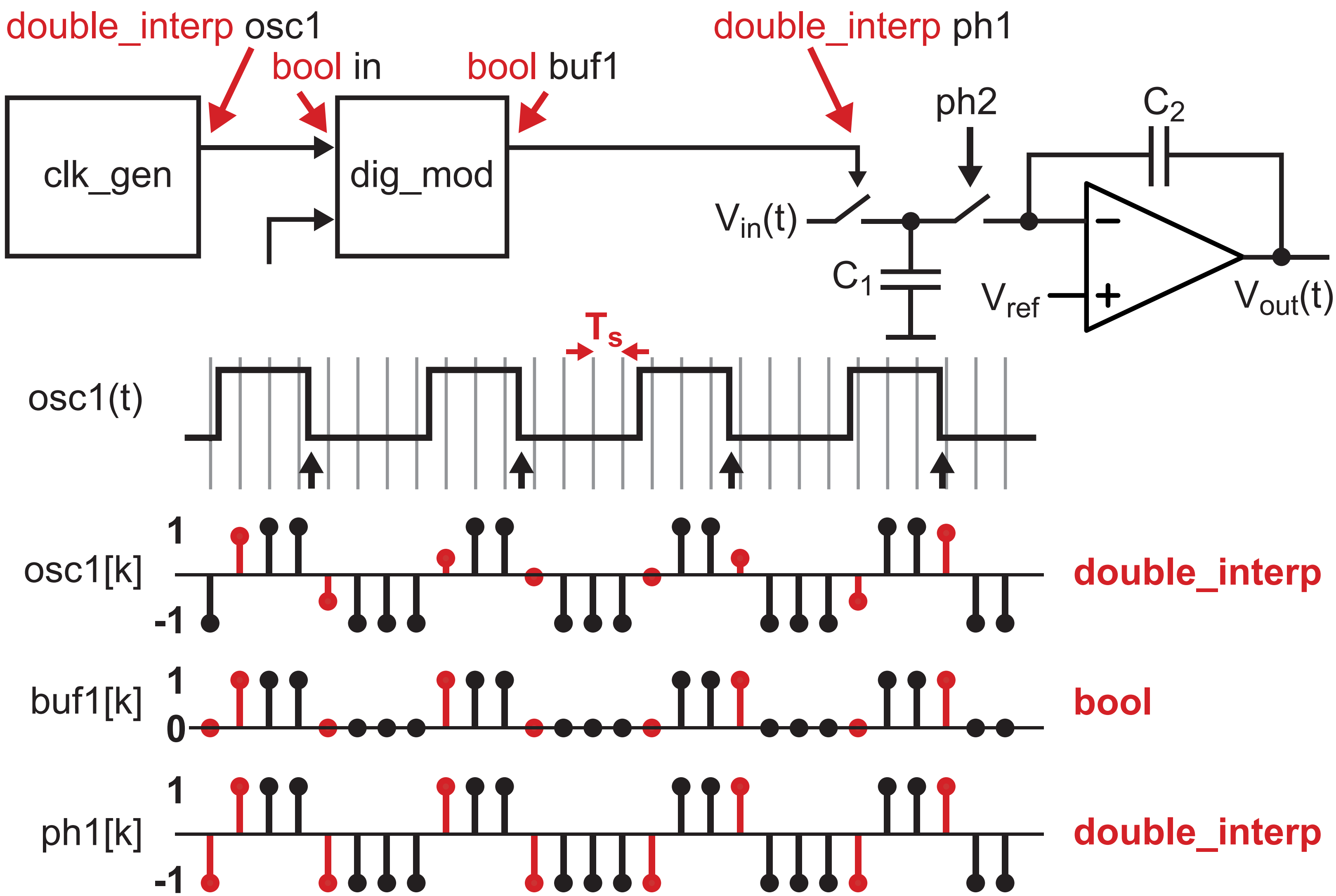
- **Electrical switches within CppSim require double_interp signals for the control nodes**
  - **Good timing accuracy achieved despite constant time step**

Feeding Bool Input with Double_Interp Signal



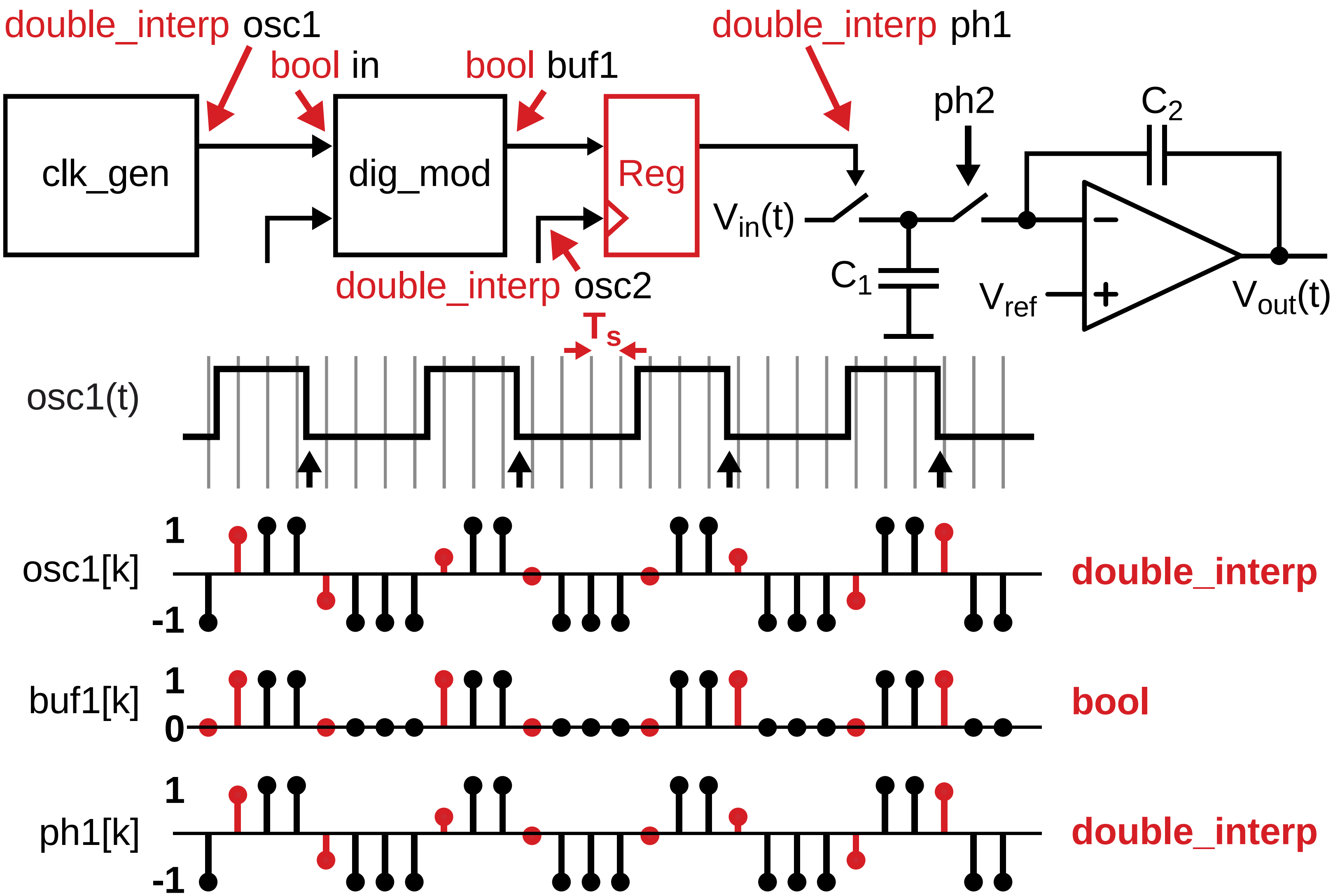
- **Conversion module automatically inserted**
  - **-1,1 signaling converted to 0,1 signaling**
  - **High resolution edge timing information is lost**

Feeding Double_Interp Input with Bool Signal



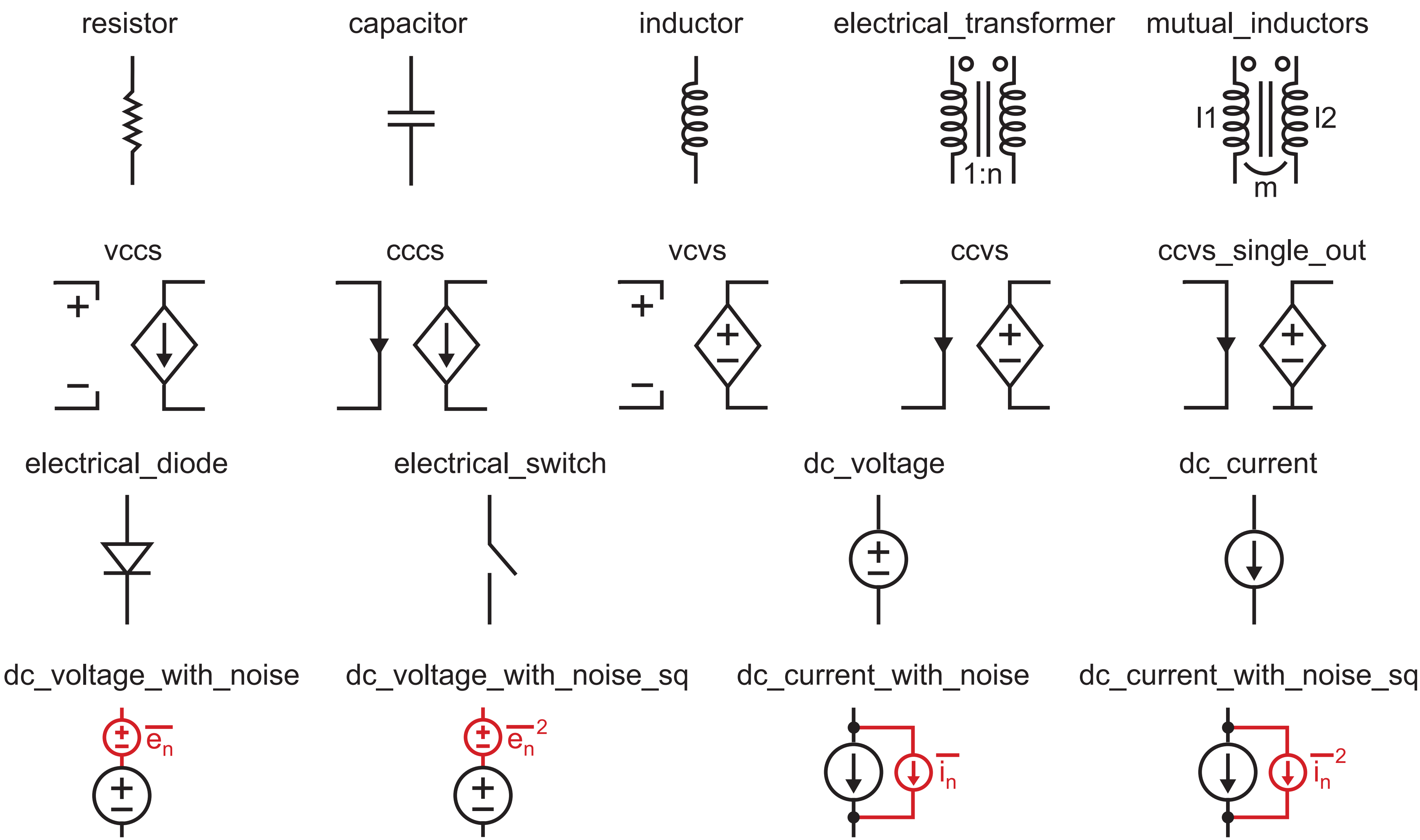
- **Automatic translation of 0,1 signaling to -1,1 signaling**
  - **Loss of timing information causes quantization noise!**

Restoring Fine Timing Information

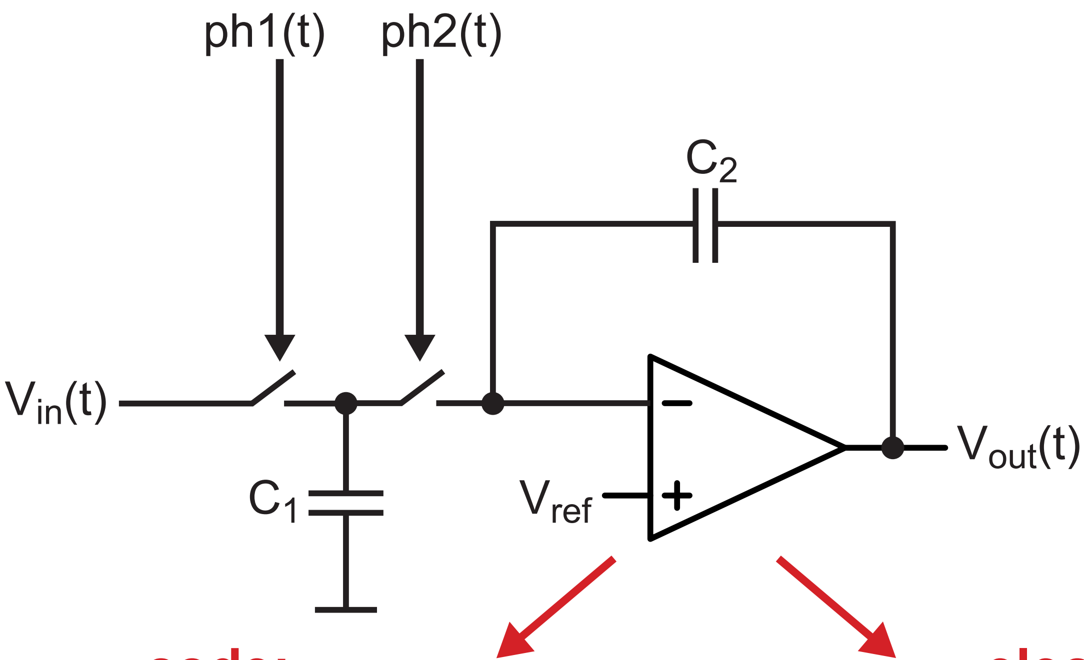


- **Use dff or reg_double_retime (Library: CppSimModules)**
  - **Above figure is simplified – ignores some additional delays**

Supported Electrical Elements in CppSim



CppSim Code Versus Electrical Element Modules



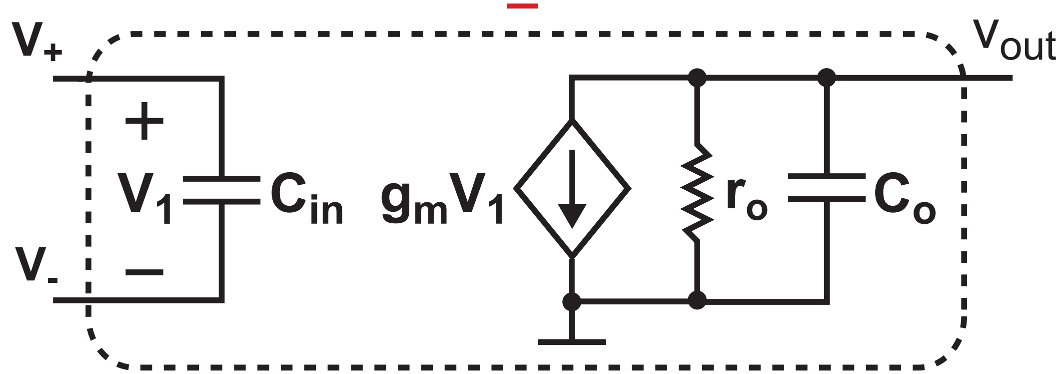
code:

Filter filt1("K","1+1/wo*s",...)

⋮

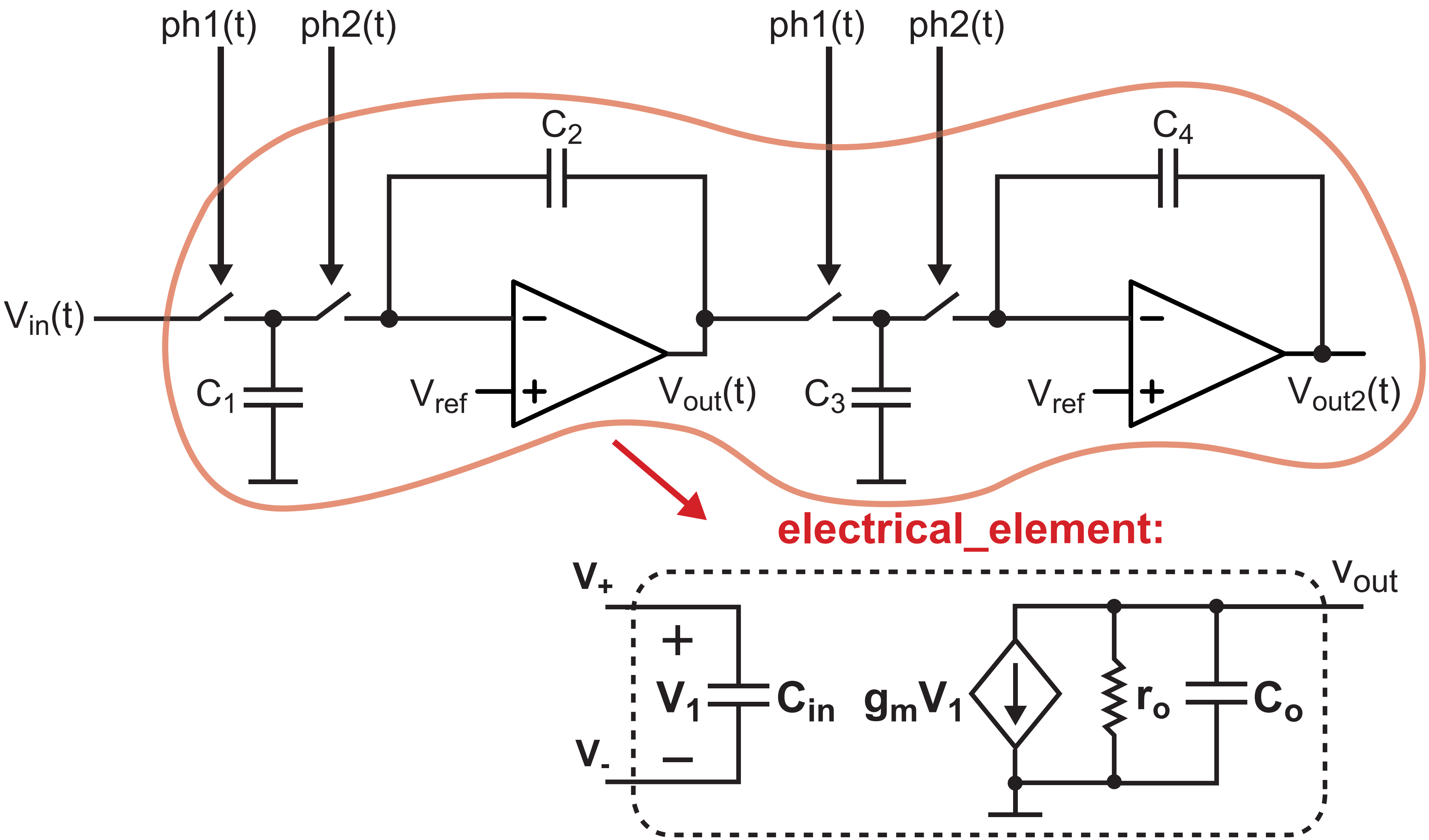
vout = filt1(vinp-vinm)

electrical_element:



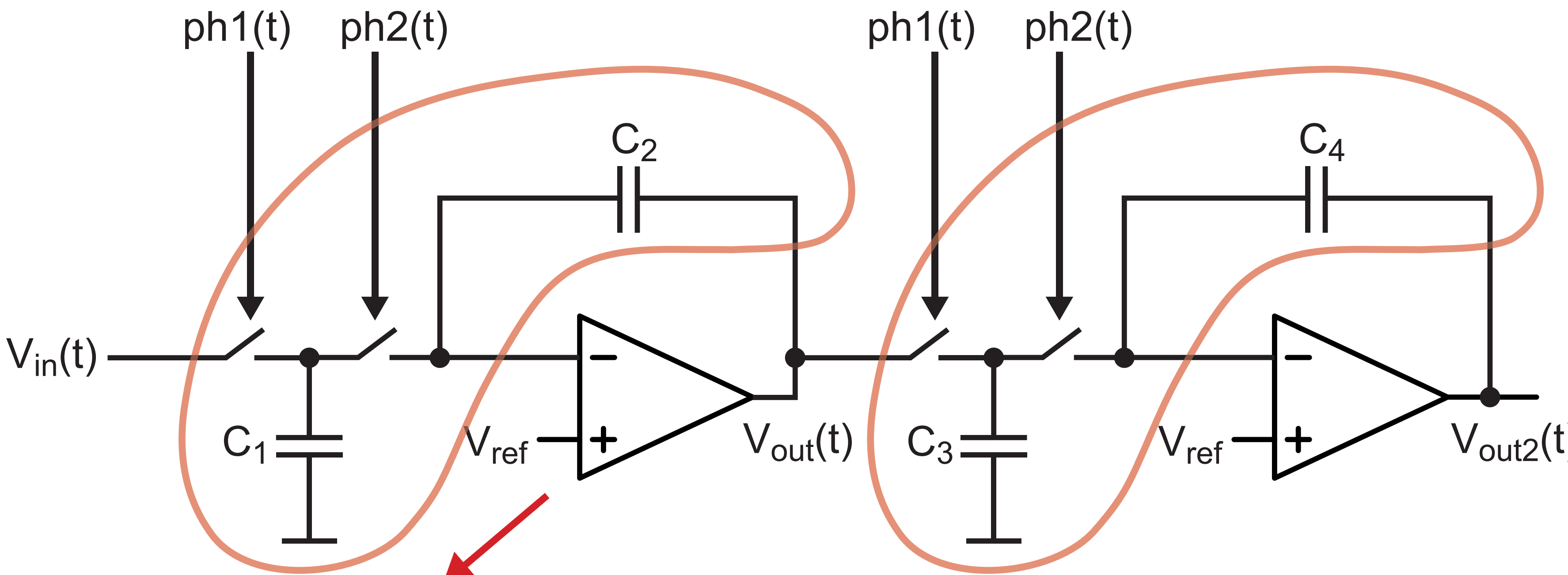
- **Which approach is best for circuit blocks such as opamps?**

Complexity Issue with Electrical Element Modules



- **State-space calculations increase as (number of nodes)^2**
  - **Large networks dramatically slow down simulation speed**

Code Modules Allow De-Coupling Between Networks



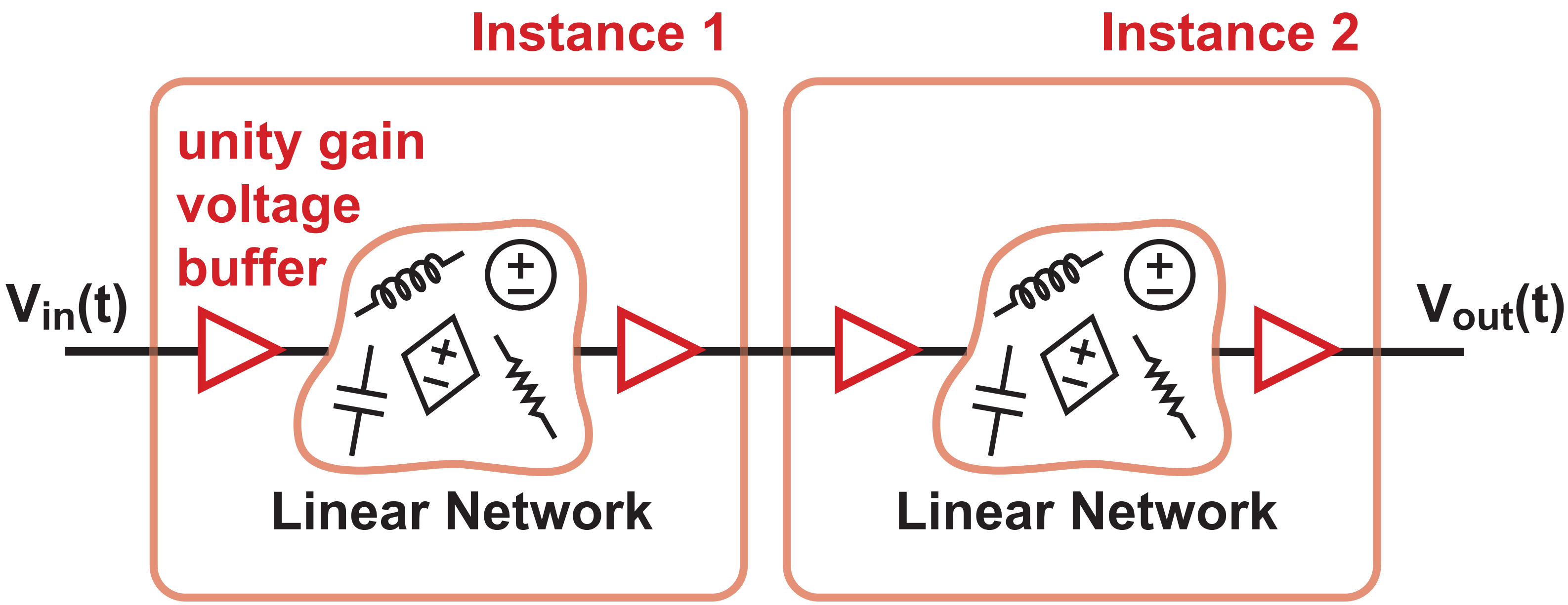
code:

```
Filter filt1("K","1+1/wo*s",...)
   ⋮
vout = filt1(vinp-vinm)
```

- **Code modules are not sensitive to loading**
  - **Allows CppSim to automatically separate into sub-networks**

Code modules preferred to achieve fast simulation speed

Impact of Hierarchy on Electrical Element Networks

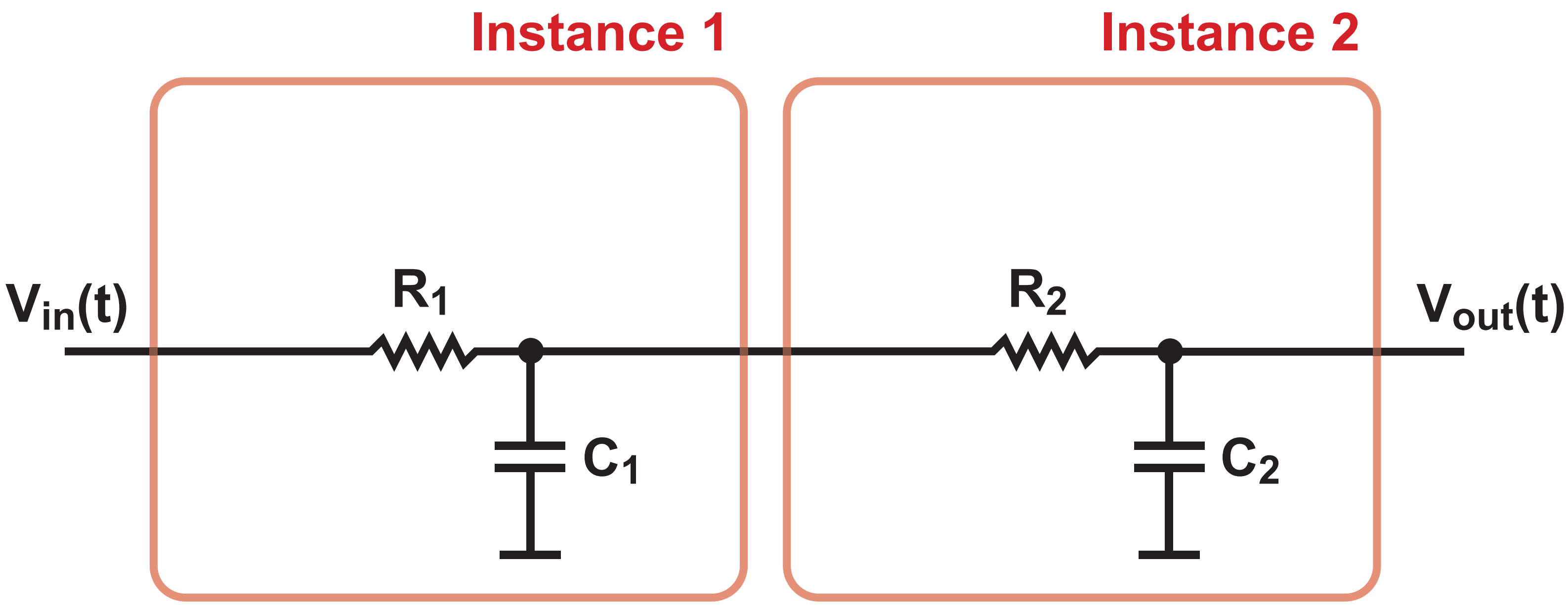


- CppSim implicitly inserts unity gain voltage buffers at all inputs and outputs of instances
  - Allows hierarchical simulation structure of overall system to be retained
  - De-couples networks at instance level to discourage creation of large state-space models

Example: A Second Order RC Network

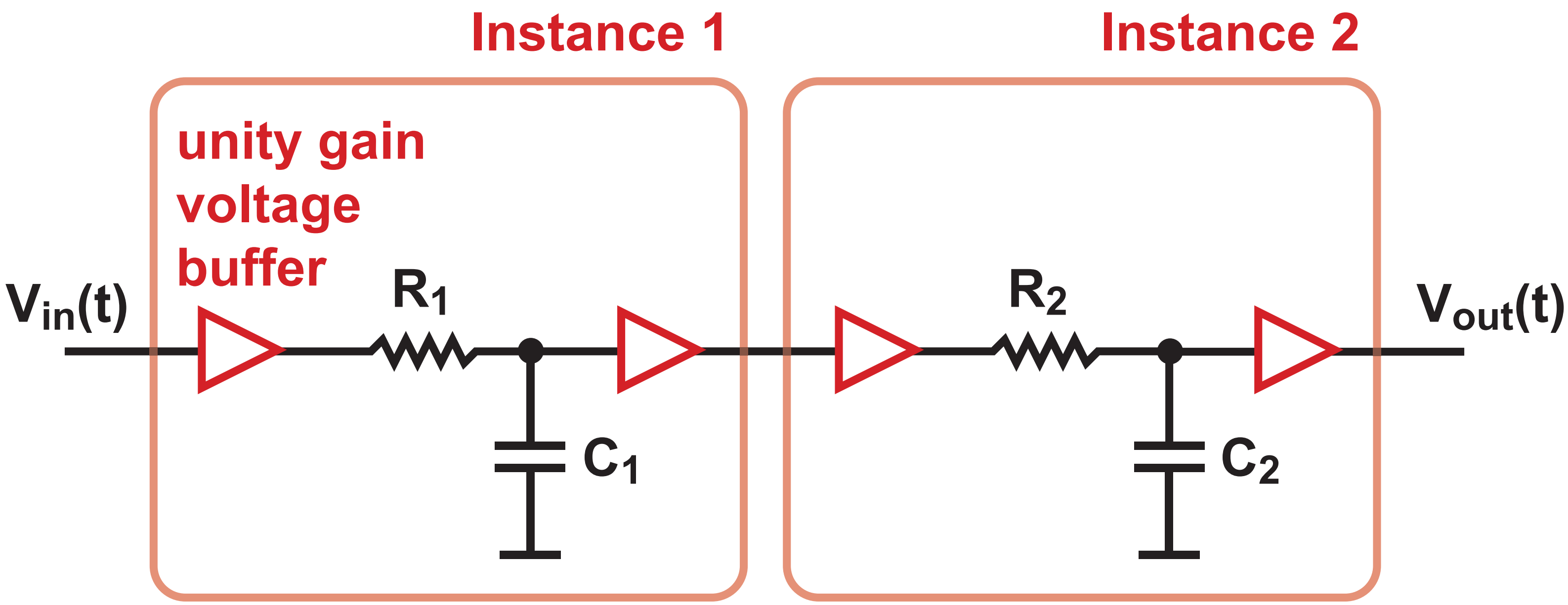
- **Resulting transfer function is *NOT* simply the cascade of two identical RC filters**
  - **Actual pole locations are influenced by mutual coupling of the two first-order RC networks**

Cascade of First Order RC Networks as Instances



- **This would appear to be the same as cascading the RC networks at the same level of hierarchy…**

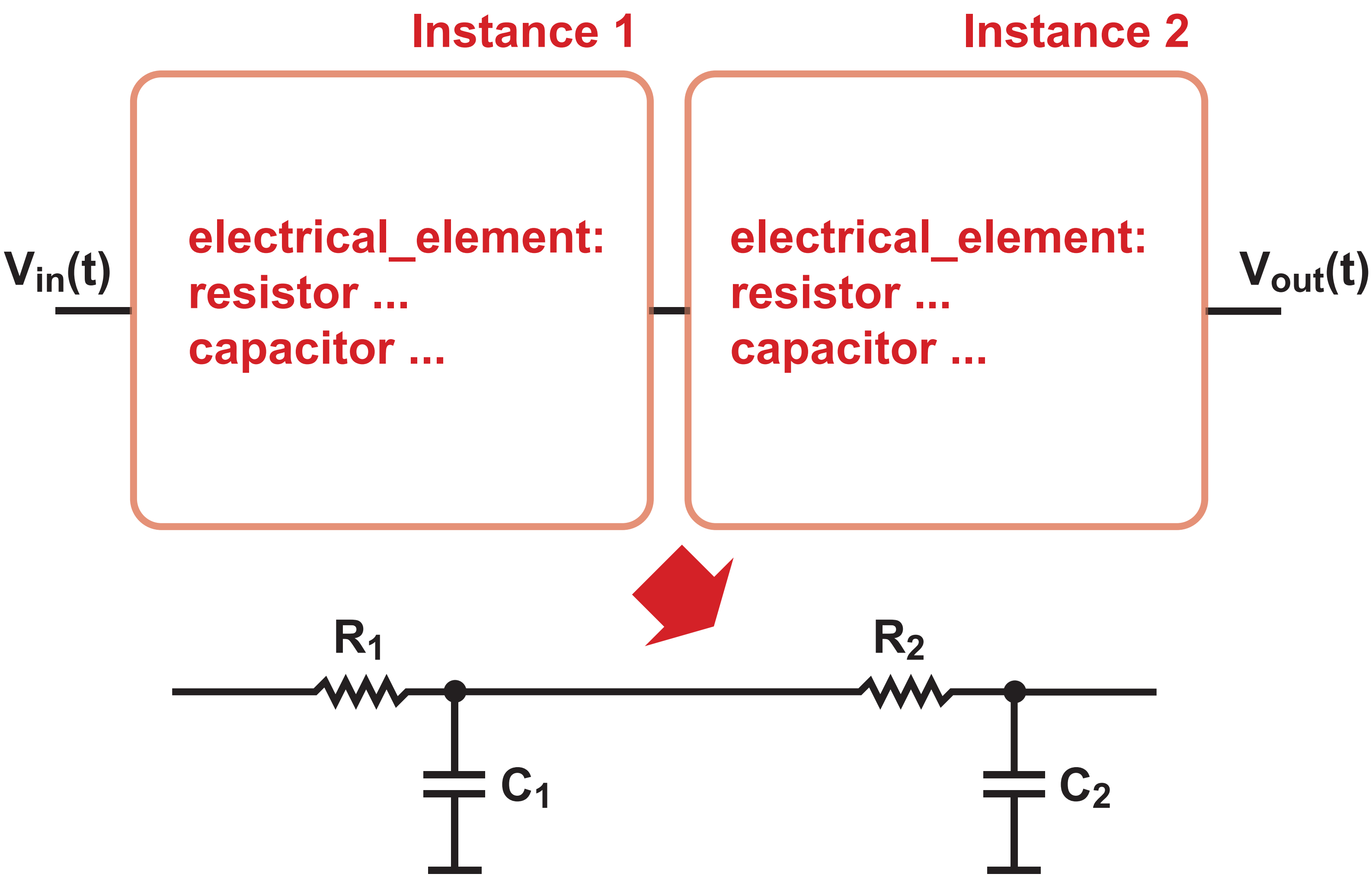
Recall Unity Gain Voltage Buffer Insertion



- CppSim implicitly adds unity gain voltage buffers
  - Resulting transfer function is actually the cascade of two identical RC filters

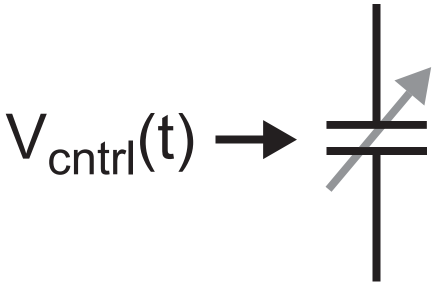
How do you achieve network coupling with hierarchy?

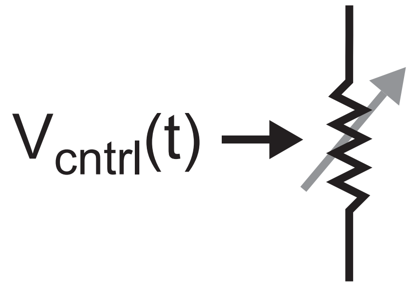
Electrical Element Modules Form Coupled Networks



CppSim allows one level of hierarchy for coupled networks

Voltage-Controlled Capacitance and Resistance





- **Electrical elements are limited to linear components**
  - **Combine CppSim modules with electrical elements to create nonlinear circuits**
    - Key technique: use CppSim module to perturb the behavior of the linear electrical element based on the voltage across its terminals and the input control voltage
- **Examples are provided of voltage-controlled capacitance and resistance in CppSim (Windows/Mac)**
  - **Library: Electrical_Examples**
    - Voltage-controlled capacitance: test_varcap_electrical
    - Voltage-controlled resistance: test_var_res_electrical

Summary of Analog Modeling in CppSim

CppSim Code Modules

- **Require unilateral flow but allow arbitrary analog functions including nonlinearity, filtering, hysteresis, etc.**

Electrical Element Modules

- **Enable straightforward modeling of linear networks with switches (and, to a more limited extent, diodes)**
  - **User simply creates schematic level representation**
  - **State-space model of network automatically created**
- **Fast speed retained by keeping network sizes small**
  - **De-coupled networks are automatically separated**
  - **Instances are decoupled unless they are electrical elements**
- **High accuracy retained for time-based circuits**
  - **Constant time step allows straightforward FFT analysis**
  - **Double_interp protocol enforced for electrical switches**

CppSim versus VppSim

- **CppSim**
  - **C++ is the simulation engine**
    - Verilog code translated into C++ classes using Verilator
  - **Best option when system simulation focuses on analog performance with digital support**
- **VppSim**
  - **Verilog is the simulation engine**
    - C++ blocks accessed through the Verilog PLI
  - **Best option when system simulation focuses on digital verification with C++ stimulus**

Constant time step approach allows seamless connection between C++ and Verilog models

VppSim Example: Utilize CppSim Module in Verilog

CppSim module

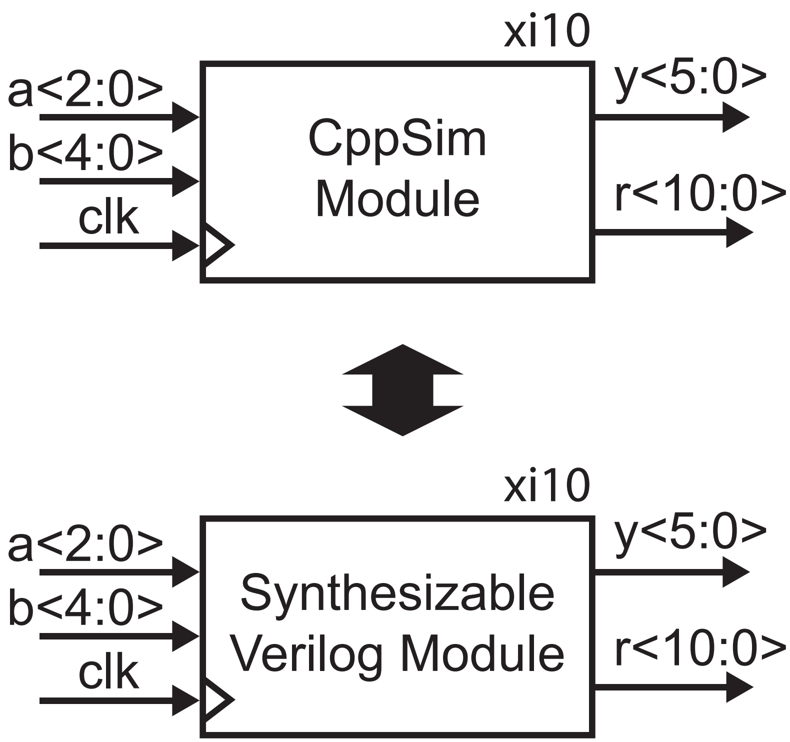
```
module: leadlagfilter
parameters: double fz, double fp,
                  double gain
inputs: double in
outputs:  double out
static_variables:
classes: Filter filt("1+1/(2*pi*fz)s",
             "C3*s + C3/(2*pi*fp)*s^2",
             "C3,fz,fp,Ts",1/gain,fz,fp,Ts);
init:
code:
filt.inp(in);
out = filt.out;
```

Resulting Verilog module for VppSim

```
////// Auto-generated from CppSim module //////
module leadlagfilter(in, out);
  parameter fz = 0.00000000e+00;
  parameter fp = 0.00000000e+00;
  parameter gain = 0.00000000e+00;
  input in;
  output out;

  wreal in;
  real in_rv;
  wreal out;
  real out_rv;

  assign out = out_rv;

  initial begin
      assign in_rv = in;
    end

  always begin
      #1
      $leadlagfilter_cpp(in_rv,out_rv,fz,fp,gain);
    end
endmodule
```

Digital Modeling in CppSim

Code Modules: CppSim or Synthesizable Verilog

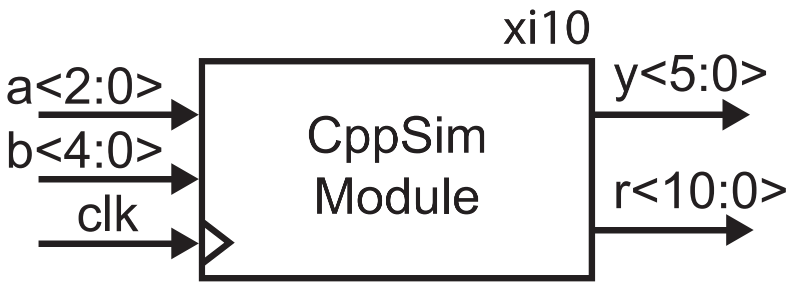



```
module: dig_mod
inputs:
bool a[2:0], bool b[4:0], bool clk
outputs:
bool y[5:0], bool r[10:0]
⋮
```

```
module dig_mod(a, b, clk, y, r);
input [2:0] a;
input [4:0] b;
input clk;
output [5:0] y;
output [10:0] r;
⋮
```

- **CppSim modules utilize bool signals**
  - **Correspond to integer vectors whose elements are 0 or 1**
- **Verilog modules must be synthesizable in CppSim**
  - **Note: full support of Verilog in VppSim**

Getting and Setting Boolean Signal Values (CppSim)

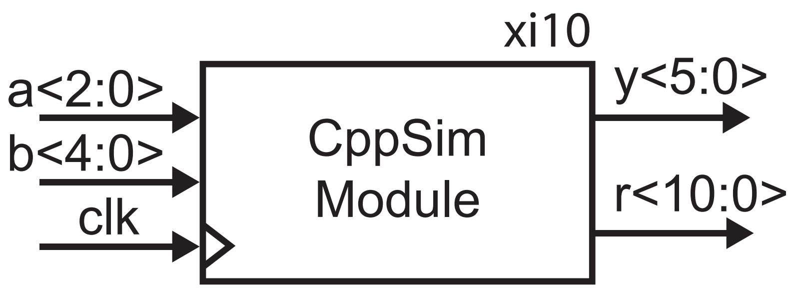


module: dig_mod
inputs:
bool a[2:0], bool b[4:0], bool clk
outputs:
bool y[5:0], bool r[10:0]
⋮

```
a_dec = a.get_decimal_value();      // full bit range (a[2:0])
b_dec = b.get_decimal_value(3,1);   // limited bit range (b[3:1])
b_bit1 = b.get_elem(1);             // get b[1]
        ⋮
y.set_decimal_value(15);            // full bit range (y[5:0] = 15)
r.set_decimal_value(21,7,2);        // limited bit range (r[7:2] = 21)
r.set_elem(8,1);                    // set r[8] = 1
```

- **Bool signals: integer vectors with element values of 0 or 1**
  - **Support functions such as get_elem(), set_elem(), etc.**
  - **For convenience: get_decimal_value(), set_decimal_value()**
    - Restricted to 32-bit values

Implementing Clock Edge Based Processing



module: dig_mod
inputs:
bool a[2:0], bool b[4:0], bool clk
outputs:
bool y[5:0], bool r[10:0]
⋮

```
EdgeDetect pos_clk_edge()
EdgeDetect neg_clk_edge()

code:
if (pos_clk_edge.inp(clk))
    {
        ⋮
    }
if (neg_clk_edge.inp(-clk))
    {
        ⋮
    }
```

```
timing_sensitivity:  posedge clk

code:
        ⋮
```

- **timing_sensitivity: clk must be of type bool**
- **EdgeDetect: clk must be of type double_interp**

EdgeDetect() versus timing_sensitivity: for VppSim

EdgeDetect (simplified)

```
////// Auto-generated from CppSim module //////
module dig_mod(a,b,clk,y,r);


  always begin
     #1
     $dig_mod_cpp(a,b,clk,y,r);
  end
endmodule
```

- PLI routine is called *every* time step
  - Dramatically slows down VppSim!

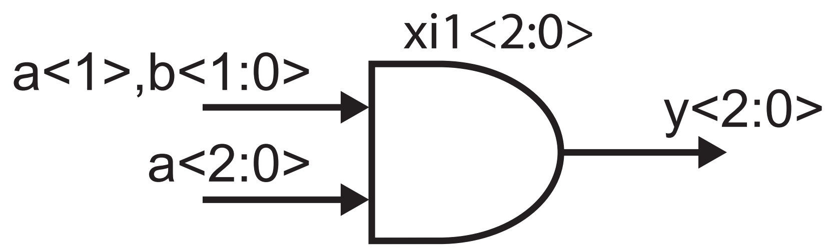
timing_sensitivity:

```
////// Auto-generated from CppSim module //////
module dig_mod(a,b,clk,y,r);


  always@(posedge clk) begin
     $dig_mod_cpp(a,b,clk,y,r);
  end

endmodule
```

- PLI routine is only called on positive clk edges
  - Much less impact on simulation speed

Use timing_sensitivity: unless you need to perform computation during every time step (Note: no penalty for EdgeDetect method in CppSim)

Buses, Bundles, and Iterated Instances



- **Basic conventions supported**
  - **Iterated instance: xi1<2:0>**
  - **Bus: a<2:0>**
  - **Bundle: a<1>,b<1:0>**
- **Key rules for bused signals:**
  - **Code modules: buses only valid for type bool**
    - Exception for electrical_element: modules:
      - Declare as bool, but actual type becomes double
  - **Schematic signals: buses can be any type**

VppSim Example: Using Buses in CppSim Module

CppSim module

```
module: queue2
parameters: int bit_width
inputs:     double_interp clk,
            double rst_n,
            bool in[2047:0],
            int enqueue,
            bool dequeue[31:0]
outputs:    bool out[2047:0],
            bool not_empty[31:0],
            int not_full

                •
                •
                •
```

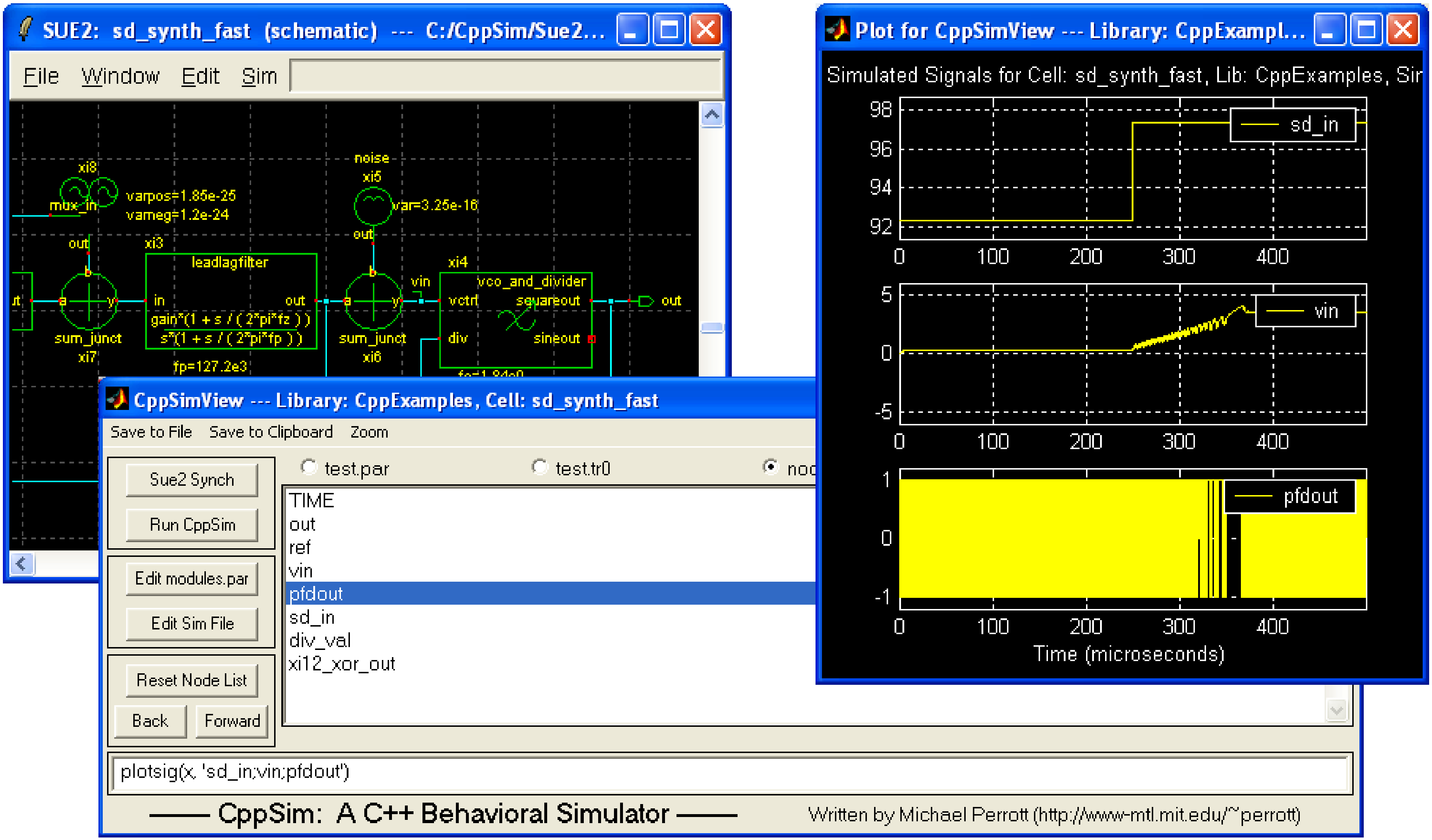
Resulting Verilog module for VppSim

```
/////////// Auto-generated from CppSim module ///////////
module queue2(clk, rst_n, in, enqueue,
                     dequeue, out, not_empty,
                     not_full);

  parameter bit_width = 0;
  input clk;
  input rst_n;
  input [2047:0] in;
  input [31:0] enqueue;
  input [31:0] dequeue;
  output [2047:0] out;
  output [31:0] not_empty;
  output [31:0] not_full;

  wreal clk;
  real clk_rv;
  wreal rst_n;
  real rst_n_rv;

                •
                •
                •
```

Summary of Digital Modeling

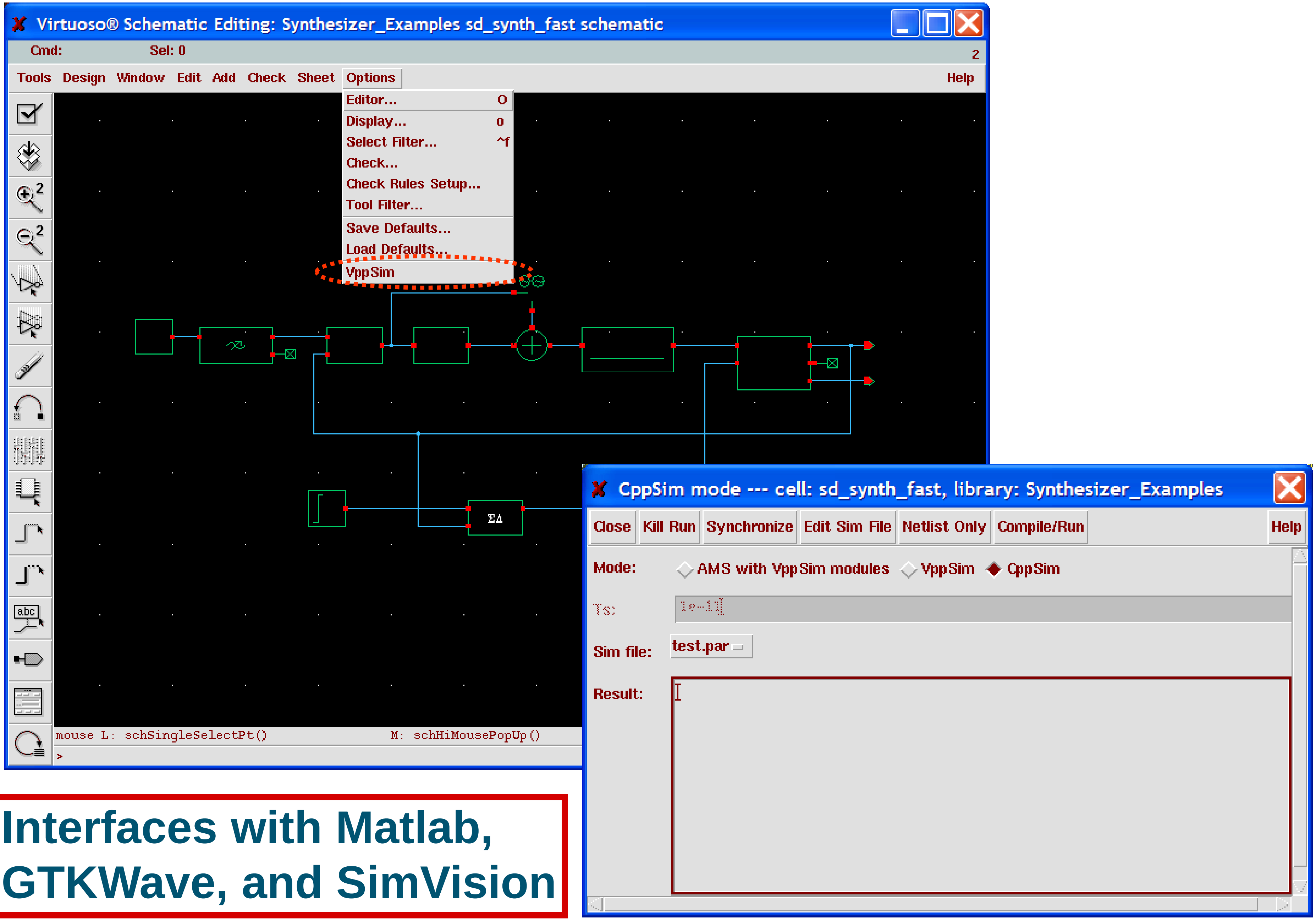
- Verilog or CppSim code modules are supported
  - CppSim simulator: Verilog must be synthesizable code
  - VppSim simulator: Verilog is fully supported
- Key constructs for CppSim modules:
  - bool signal type allows bus notation
  - timing_sensitivity: advantageous for VppSim simulator
- Buses, bundles, and iterated instances supported
- Care should be taken to avoid introducing timing quantization noise when passing digital signals back to analog
  - Conversion of double_interp signals to type bool leads to loss of high resolution timing information of edges

Screenshot of CppSim/VppSim (Windows Version)



Readily Interfaces with Matlab and GTKWave

Screenshot of CppSim/VppSim (Cadence Version)



Interfaces with Matlab, GTKWave, and SimVision

Free Download at www.cppsim.com

Many Tutorials Available for CppSim/VppSim

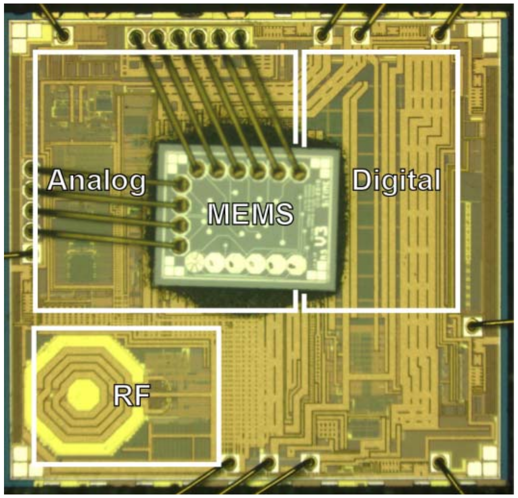
- **Switched Capacitor 2nd Order Delta-Sigma ADC**
- **Phase Locked Loops (Analog and Digital)**
- **VCO-based ADCs**
- **GMSK modulator**
- **Decision Feedback Equalization**
- **Optical-Electrical Downversion and Digitization**
- **OFDM Transceiver**

➡ **See http://www.cppsim.com**

Example Benchmarks for a Full Chip Simulation

Tabulated simulation times for a MEMS-based oscillator:

- **SPICE-level model**
  - **Checking of floating gate, over-voltage, startup of bandgap and regulators, etc.**
    - **Spectre Turbo: 2 microseconds/day**
    - **BDA: 8 microseconds/day**
- **Architectural model using CppSim**
  - **Examination of noise and analog dynamics**
    - **2.8 milliseconds/hour**
- **Verification model using VppSim**
  - **Validation of digital functionality in the context of analog control and hybrid digital/analog systems**
    - **7 milliseconds/minute**



Conclusion

- **CppSim is designed for high productivity and versatility**
  - **Easy to create your own code blocks**
    - Use existing modules to see examples, but don’t limit yourself to what is available
  - **Allows very detailed modeling of complex circuits**
    - You are not confined to an overly simplified model
  - **Invites a scripted approach to running simulations**
    - Excellent integration with Matlab/Octave and Python
  - **Runs in Windows, Mac OS X, or within Cadence**
    - Has been used to simulate entire ICs in Cadence

- **Extensive 14 year track record of enabling new circuit architectures with first chip success**