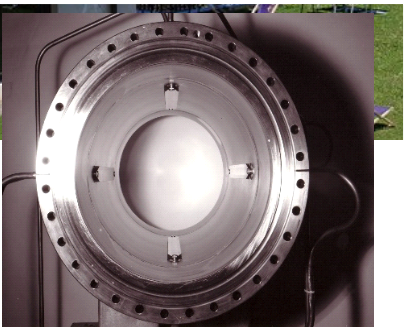
The **CERN Accelerator School**

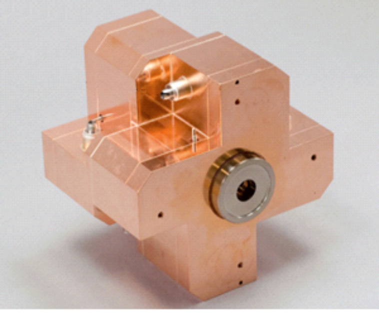
ADVANCED ACCELERATOR PHYSICS

The CERN Accelerator School

06 – 18 November 2022

Neaclub, Sévrier, France





AFTERNOON COURSE on Beam Position Measurement

- Introduction to the Simulation Tool
- Understanding Beam Position Monitors
- Beam Position Acquisition Systems
  - Analogue to Digital Conversion
  - Homodyne Receivers

Marek Gasior & Rhodri Jones (CERN)

CERN Accelerator School
CH – 1211 Geneva 23
cas.web.cern.ch
Accelerator.school@cern.ch

Introduction to the Course on Beam Position Measurement
CERN Advanced Accelerator School 2022

M. Gasior & R. Jones (CERN)
Sevrier, France. 6th – 18th November 2022

Purpose

The aim of this course is to give you a basic understanding of the measurement principles, fundamental concepts and related technological aspects of deriving beam position in a particle accelerator. An interactive simulation tool will allow you to generate beam signals and construct your own virtual monitor and acquisition systems. The simulation course will be complemented by a practical laboratory session, where you will be able to physically measure what you've been simulating, to further enhance your understanding of the concepts behind such systems.

Introduction to the Simulation Tool:

1. The Workspace

The simulation tool is based on a main workspace (Fig. 1) where the components to be simulated are put by selecting the appropriate circuit drawing block. The blocks are connected using wires, with the signals on any wire capable of being probed. The *Circuit Drawing* **Block** contains most of the standard components required to simulate a beam and build a beam position monitor acquisition system.

Beam:
Revolution Period (Trev): 100 ns
Revolution Frequency: 10 MHz
RF Bucket Length: 100 ns
Harmonic Number: 1

Simulator Status:
Step: 40 ps
Length: 1 * [Trev]
Points per RF Bucket: 2.5 k
Total Steps: 2.5 k

Simulation Parameters
Points per Bucket: 100 ☑ Auto
No. of Revolutions: 1 ☑ Auto

Simulaton Data:
☑ Save Simulations
☐ Suppress Warnings
Simulation Run: 0

Operation Mode: Normal Mode

Fig. 1: Workspace with circuit drawing *Block* open

1.1 Connecting Items

Once selected and dropped onto the simulator workspace the output from one ITEM can be provided to the input of another ITEM by clicking on **Place Wire** in the *Circuit Drawing* menu. Clicking on an input or output node (small icon rectangles within each ITEM) then generates a wire, which can be terminated by clicking on another input or output node. REMEMBER to click on **Exit Place Wire** when you’ve finished with the wiring.

1.2 Probing Signals

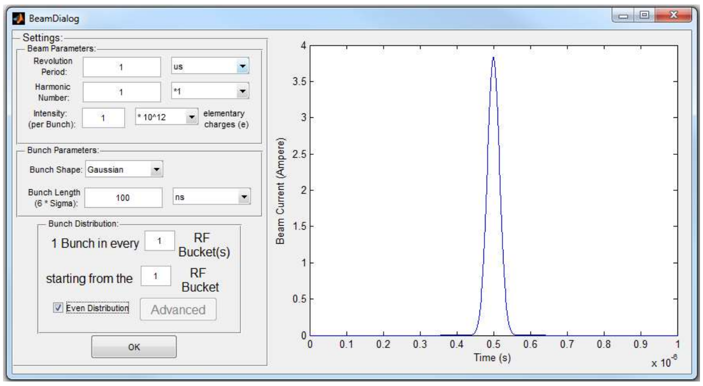
To look at a signal at any point in the acquisition chain requires a probe to be placed on a wire (or a wire to be joined to a probe). Select **Place Probe** from the *Circuit Drawing* menu & click on the wire to be probed, or drop the probe with a left click & attach the automatically generated wire to the point to be probed.

1.3 Saving and Loading Circuits in the Workspace

A circuit can be saved at any time by clicking on the **Save** button in the *Circuit* menu. This will save the circuit and the parameters currently attributed to all components. A saved circuit can be retrieved using the **Load** button in the *Circuit* menu.

2. Component Properties

Once selected and dropped onto the simulator workspace the properties of each component can be modified by right clicking on the item and selecting *“properties”* from the dropdown menu. For example the beam itself is represented by the *“Beam”* component in the *“Sources”* section of the circuit drawing ***Block*** menu. This appears as ITEM1 in the workspace shown in Fig. 1. The *“properties”* of this component (Fig. 2) allows the user to set all the parameters necessary to simulate most standard beam types.



BeamDialog

Settings:

Beam Parameters:

Revolution Period: 1 us

Harmonic Number: 1 *1

Intensity: (per Bunch): 1 * 10^12 elementary charges (e)

Bunch Parameters:

Bunch Shape: Gaussian

Bunch Length (6 * Sigma): 100 ns

Bunch Distribution:

1 Bunch in every 1 RF Bucket(s)

starting from the 1 RF Bucket

☑ Even Distribution

Fig. 2: BeamDialog showing the current properties of the BEAM generator component

In the example of Fig. 2 the beam is defined to be a single Gaussian bunch of length (6 x sigma) = 100ns containing 10^{12} charges. The revolution period is 1 microsecond (revolution frequency of 1MHz), with a harmonic number (number of RF buckets per revolution period) of 1, corresponding to an RF frequency which is also 1MHz. For higher harmonic numbers it is possible to further define how many bunches there are per turn and where these bunches are positioned, using the Bunch Distribution panel which can be opened by unselecting the *“Even Distribution”* checkbox.

All the ITEMs in the Circ*uit Drawing* **Block** will open similar property windows that allow the user to define their parameters.

3. Simulating the Circuit

Once you're happy with the model, and have placed probes at all locations that you wish to monitor, the simulation can be launched. To do this press the **Simulate** button in the *Simulate Actions* menu. Once the simulation is complete you will be asked to name the output data before the Results Viewer (Fig. 3) is automatically opened to show the results.

4. Displaying Results

The Results Viewer is comprised of two graphs, the top graph showing the time evolution of the signal at each probe and the lower graph showing the frequency content of the signal at each probe (i.e. the signal magnitude spectrum calculated from the Fourier Transform of the time domain data).

Each time you simulate the new results will be added to the existing graphs. This allows you to compare results from circuits with different settings.

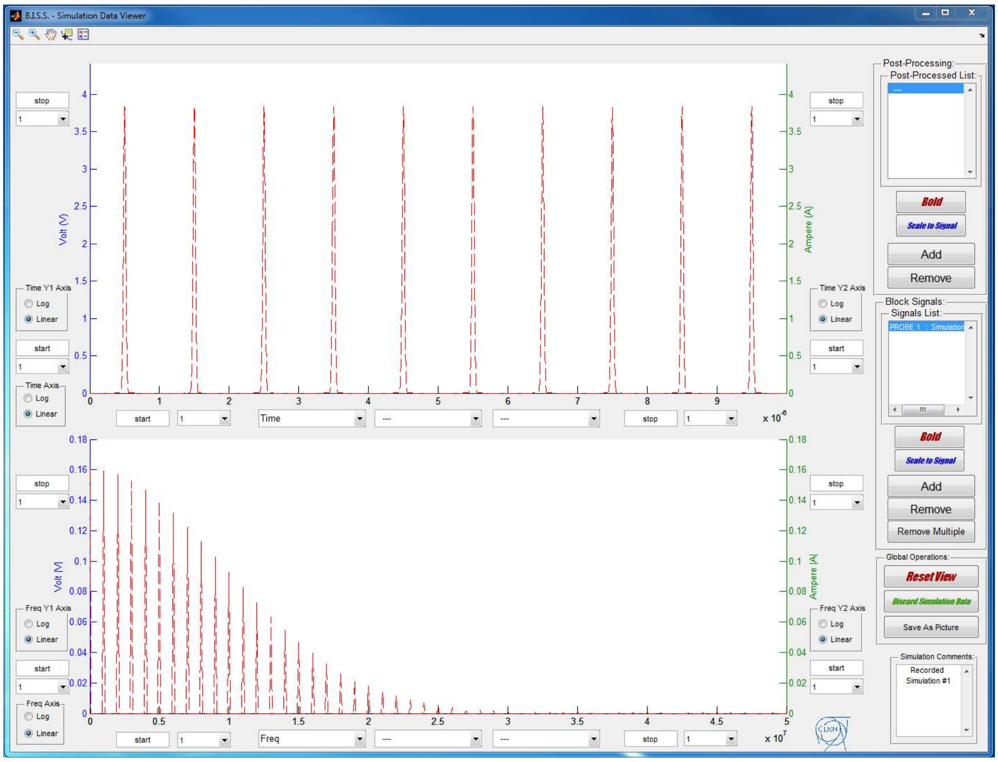


Fig. 3: The Results Viewer. Top plot shows the time evolution of the signal at a probe, while bottom plot shows the frequency content of the signal for the same probe.

5. Manipulating the Graphs in the Results Viewer

5.1 Global Figure Functions

Zooming, obtaining data point values and adding or removing the legend can be performed using the top menu bar in the Results Viewer.

5.2 Axis properties

The axis scale for each graph can be set manually if desired by replacing the "start" or "stop" with a value and selecting the appropriate scaling factor in the adjacent drop-down menu. For example in

Fig. 3 if we wish to look only at frequencies up to 20MHz, then “stop” should be replaced by 20 with 10^6 selected in the adjacent drop-down menu. It is also possible to display the data on a logarithmic scale by selecting the appropriate check-box. Clicking on **Reset View** will restore the original, auto-scaled display.

5.3 Highlighting Traces

When there are multiple traces present in the Results Viewer it is possible to highlight one of the traces for easier viewing. First select the trace to be highlighted from the **Signal List** in the *Block Signals* menu and then click on the **Bold** button.

5.4 Discarding Traces

To discard simulation results simply select the **Discard Simulation Data** button in the *Global Operations* panel of the Results Viewer or the *Simulate Actions* menu of the workspace. Select the trace or traces (result data) you wish to supress from the list and click OK.

5.5 Saving Results

In order to save a screenshots of a graph to file, first click on the graph to be saved and then click on the **Save as Picture** button in the *Global Operations* panel. Please note that as only one graph is saved at a time it is important to first select the graph to be saved by clicking on it before clicking on the **Save as Picture** button.

5.6 Post-Processing Results

If you wish to perform simple mathematical operations on the data, or combine signals and then display the results, this is possible using the **Add** button in the *Post-Processed List* panel. A ComplexSignalDialogue window will then open to allows mathematical functions to be defined for the new, post-processed trace. This allows two or more traces to be combined via simple addition, subtraction, multiplication or division. It also gives the option of producing a trace of the integrated signal or to find the average, rms, standard deviation, maximum value, minimum value or peak-peak value for any existing traces.

Course on Beam Position Measurement
CERN Advanced Accelerator School 2022

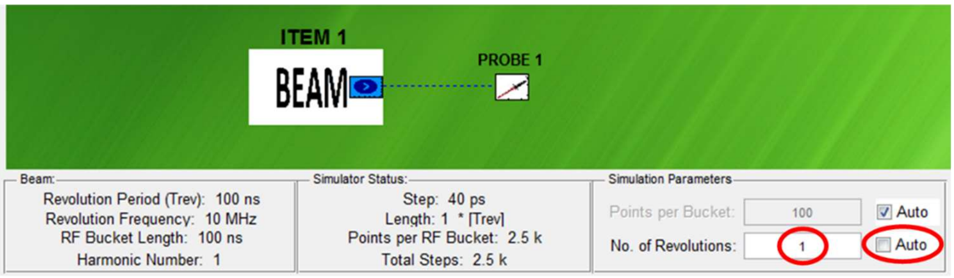
Understanding Beam Position Monitors:

1. Beam Signals in the Time and Frequency Domain

In order to be able to correctly design a pick-up system, it is important to be able to switch between the time and frequency domain. Let's therefore first try to understand what the signal from a Beam would look like if we were able to directly measure it with an ammeter.

1) Let's assume we have a single Gaussian bunch of length (6σ) = 2ns containing 5×10^9 charges. We'll set the Revolution Period to 100ns (revolution frequency of 10MHz) and the number of turns to 1, so that we can see what the signal from a single passage of this bunch looks like.

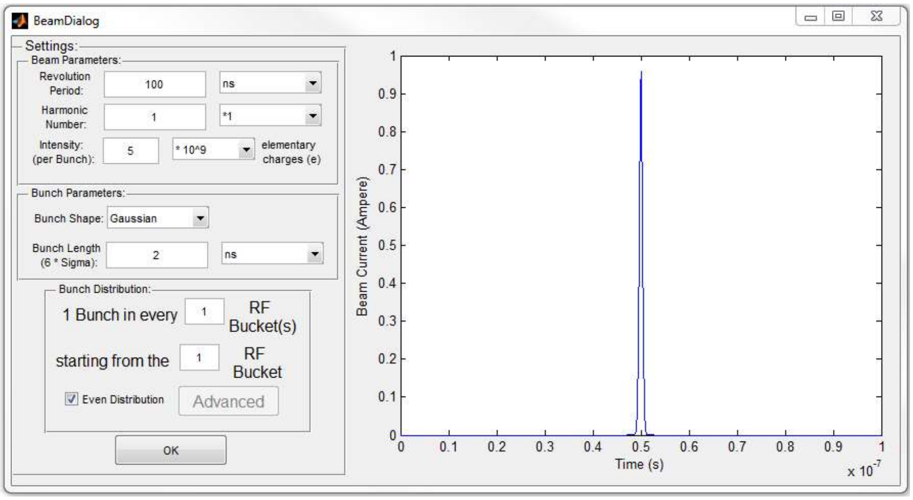
- *Using the* ***Circuit Drawing*** ⇒ ***Block*** ⇒ ***Sources*** ⇒ ***BEAM*** *module produce the time and frequency domain plot for this beam, with the following settings:*



Beam:
Revolution Period (Trev): 100 ns
Revolution Frequency: 10 MHz
RF Bucket Length: 100 ns
Harmonic Number: 1

Simulator Status:
Step: 40 ps
Length: 1 * [Trev]
Points per RF Bucket: 2.5 k
Total Steps: 2.5 k

Simulation Parameters
Points per Bucket: 100 ☑ Auto
No. of Revolutions: 1 ☐ Auto



Settings:
Beam Parameters:
Revolution Period: 100 ns
Harmonic Number: 1 *1
Intensity: (per Bunch): 5 * 10^9 elementary charges (e)

Bunch Parameters:
Bunch Shape: Gaussian
Bunch Length (6 * Sigma): 2 ns

Bunch Distribution:
1 Bunch in every 1 RF Bucket(s)
starting from the 1 RF Bucket
☑ Even Distribution

[N.B. Right clicking the mouse on any ITEM in the workspace opens a dialogue window where its "properties" can be found]

a) ***What can you say about the frequency content of a Gaussian input signal***

b) ***What happens to the frequency content of the signal if we increase the bunch length to 4ns or reduce it to 1ns?***

2) Although Gaussian approximations are often used in simulating beam parameters, the bunches in a real accelerator are rarely Gaussian in shape. In order to see the effect of non-Gaussian shapes on the beam spectrum:

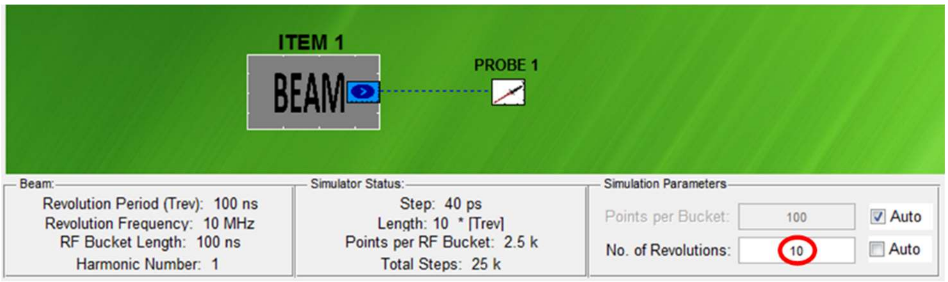
- *Discard the simulations for 1ns and 4ns, keeping only the results for 2ns (or discard all and re-simulate for a Gaussian bunch of bunch length 2ns.*
- *Set the **"Bunch Shape"** to cos^2 and **"Bunch Length"** to 2ns in the **BEAM** block.*
- *Re-simulate.*

a) ***What is the main difference?*** (hint – a log scale in frequency domain can help)

b) ***Can you think of reasons why can it can be important to take the exact shape into account?***

3) In a circular accelerator or storage ring a bunch will pass by our observation position once every turn. Let's see what happens to the bunch spectrum when have such a circulating bunch.

- *Discard all simulation results.*
- *Reproduce the simulation of a single, 2ns, Gaussian bunch for a single turn as in step 1).*
- *Now increase the simulation length to 10 turns (**Bottom Bar ⇒ Simulation Parameters ⇒ Simulation Length = 10 turns**).*



Beam:
Revolution Period (Trev): 100 ns
Revolution Frequency: 10 MHz
RF Bucket Length: 100 ns
Harmonic Number: 1

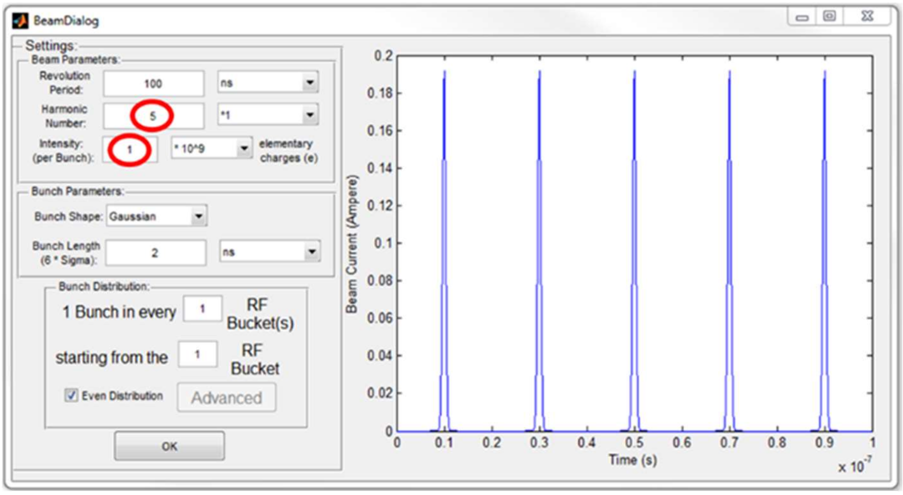
Simulator Status:
Step: 40 ps
Length: 10 * [Trev]
Points per RF Bucket: 2.5 k
Total Steps: 25 k

Simulation Parameters
Points per Bucket: 100 ☑ Auto
No. of Revolutions: 10 ☐ Auto

a) ***What is the result and how is it linked to the revolution period?***

4) Now let's consider the case of a typical machine where the RF frequency is such as to allow several bunches to be present in the machine at the same time.

- *In the **BEAM** block properties set the harmonic number = 5*
- *In the **BEAM** block properties set the intensity to 1×10^9 charges to keep the total intensity per turn constant.*
- *Re-simulate for 10 turns.*



Settings:
Beam Parameters:
Revolution Period: 100 ns
Harmonic Number: 5 *1
Intensity: (per Bunch): 1 * 10^9 elementary charges (e)
Bunch Parameters:
Bunch Shape: Gaussian
Bunch Length (6 * Sigma): 2 ns
Bunch Distribution:
1 Bunch in every 1 RF Bucket(s)
starting from the 1 RF Bucket
☑ Even Distribution

- ***a)*** ***What is now the corresponding RF frequency?***
- ***b)*** ***What happens to the frequency spectrum?***
- ***c)*** ***Note down a frequency that has:***
  - ***signal associated with it only for single bunch operation.***
  - ***signal associated with it for both single and multi-bunch operation.***

2. Beam Position Monitors

The basic quantity required to estimate the signal produced by a BPM is the transfer impedance Z_t. This is a function of frequency and connects the beam current I_{beam} with the useable output voltage U_{signal}, i.e. it tells you what output voltage you will get from your beam position monitor electrode for a given beam current as a function of frequency.

$$\mathbf{U_{signal}(f) = Z_t(f).\ I_{beam}(f)}$$

When designing a BPM system it is important to match the monitor and acquisition system to the beam characteristics – intensity, bunch length and repetition rate. Let us therefore first look at what influences the transfer impedance Z_t of a typical capacitive BPM such as the so-called button pick-up. A button pick-up is usually constructed from 4 equal round disks positioned at the top, bottom, left & right of the vacuum chamber. The transfer impedance at high frequency and the low frequency cut off for such a beam position monitor is given by:

$$Z_{T\infty} = \frac{A}{(2\pi r)\times c\times C_e} \qquad f_L = \frac{1}{2\pi R C_e}$$

Where A is the button area, 2πr the beam pipe circumference, c the speed of light, C_e the button capacitance and R the termination resistance.

(The low frequency cut-off is quoted for a difference of -3dB, i.e. the frequency at which the output voltage is a factor 1.4 lower than the maximum. Difference in dB = 20 × log × [Output Voltage/Reference Voltage])

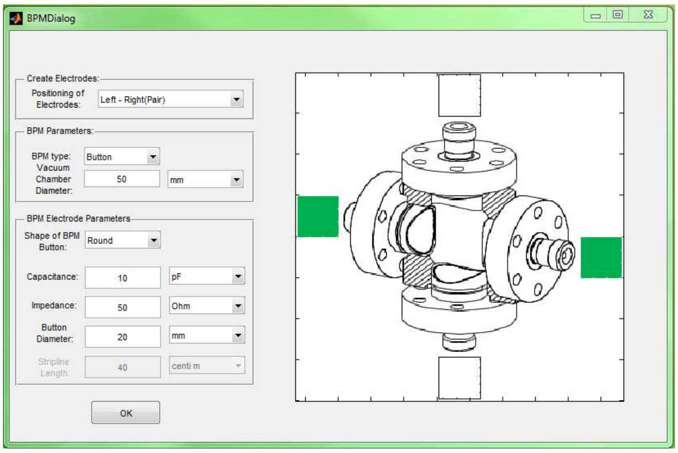
Now let's see what this means in practice.

5) *Add the **"Beam Position Monitor"** Block to the workspace and right click on it, selecting **"properties"** to get a list of the BPM parameters. To start with we'll consider measuring only in the horizontal plane, so select **"Left-Right (Pair)"** for the positioning of the electrodes. The other parameters should be as shown below and correspond roughly to an LHC type BPM:*

Beam:
Revolution Period (Trev): 100 ns
Revolution Frequency: 10 MHz
RF Bucket Length: 100 ns
Harmonic Number: 1

Simulator Status:
Step: 2 ps
Length: 1 * [Trev]
Points per RF Bucket: 50 k
Total Steps: 50 k

Simulation Parameters
Points per Bucket: 100 ☑ Auto
No. of Revolutions: 1 ☐ Auto


BPMDialog

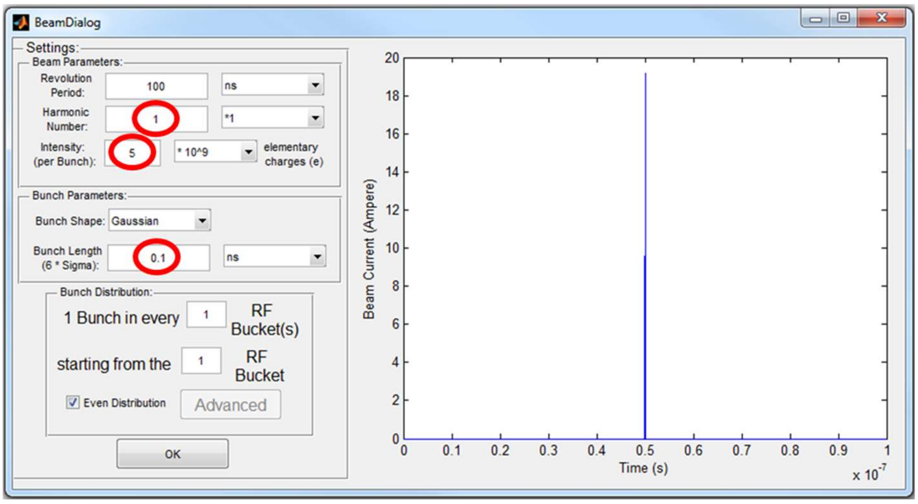
Create Electrodes:
Positioning of Electrodes: Left - Right(Pair)

BPM Parameters:
BPM type: Button
Vacuum Chamber Diameter: 50 mm

BPM Electrode Parameters
Shape of BPM Button: Round
Capacitance: 10 pF
Impedance: 50 Ohm
Button Diameter: 20 mm
Stripline Length: 40 centi m

In order to see what the response of a single button is we need to look at the so-called impulse response of the system – i.e. how the button reacts to a very short pulse.

*To do this, connect the **BEAM** module with the following settings:*


BeamDialog

Settings:
Beam Parameters:
Revolution Period: 100 ns
Harmonic Number: 1 *1
Intensity: (per Bunch): 5 * 10^9 elementary charges (e)

Bunch Parameters:
Bunch Shape: Gaussian
Bunch Length (6 * Sigma): 0.1 ns

Bunch Distribution:
1 Bunch in every 1 RF Bucket(s)
starting from the 1 RF Bucket
☑ Even Distribution

Now put a probe on one of the BPM outputs & simulate the impulse response (you will get an error message to say that the pulse is much shorter than the button and that the simulation may not be accurate – but just click CONTINUE as it's good enough for us!).

The frequency range of interest is up to some 3GHz, so set the maximum of the frequency axis accordingly. You will also need to force the secondary y-axis (in Amperes) to start at zero.

a) How would you describe the response of a button electrode to an impulse response?

b) How do the following affect the output in terms of amplitude and signal shape?

  i. Area of button?
  ii. Button Capacitance?
  iii. Termination resistance?

As the beam input can remain the same for all these simulations you can remove Probe 1 if you wish, leaving only the probe after the BPM.

6) Instead of a very short pulse, let's now put in a typical LHC type bunch length (~2ns).

- *In the **BEAM** block properties set the bunch length to 2ns*
- *Re-simulate (remember to put back a probe after the **BEAM** block if not already present)*

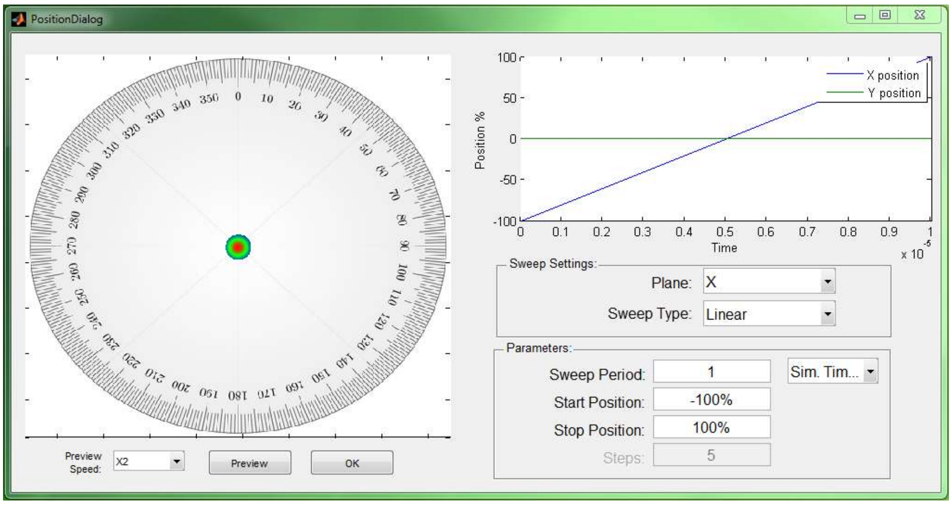
a) Compare this with the impulse response of question 5. What do you notice?

7) Using the knowledge gained in question 6, construct a button electrode with impedance 50Ω capable of providing 100mV peak signal at its output for a 2ns bunch of 5x10^9 charges.

a) What are the parameters of your button BPM?

b) At which frequency does it have the highest output signal?

8) To calculate the position using a real BPM it is necessary to compare the difference in electrode signals from opposite electrodes. In the simulation tool the beam position is controlled by the ***"POSITION"*** block.

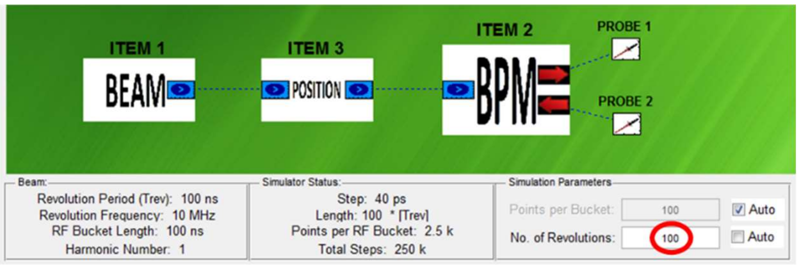


Sweep Settings:
Plane: X
Sweep Type: Linear

Parameters:
Sweep Period: 1 Sim. Tim...
Start Position: -100%
Stop Position: 100%
Steps: 5

In the example above the properties of the position block are adjusted so as to sweep the position linearly (*"Linear"* Sweep Type) in the horizontal plane (*"X"* Plane) from left to right (*"Start Position"* -100%, *"Stop Position"* 100%) over the whole aperture. The time it takes to do this is set to be the *"Sim. TimeSpan"*, i.e. the whole of the simulation time. For a simulation of 100 turns this then gives a position change of 2% of the aperture per turn.

*Construct a beam position monitor based on the button electrode you designed in 3) inserting the **"POSITION"** block in between the **"BEAM"** and **"BPM"** blocks. Probe the two outputs (left and right electrode) of the BPM.*



Beam:
Revolution Period (Trev): 100 ns
Revolution Frequency: 10 MHz
RF Bucket Length: 100 ns
Harmonic Number: 1

Simulator Status:
Step: 40 ps
Length: 100 * [Trev]
Points per RF Bucket: 2.5 k
Total Steps: 250 k

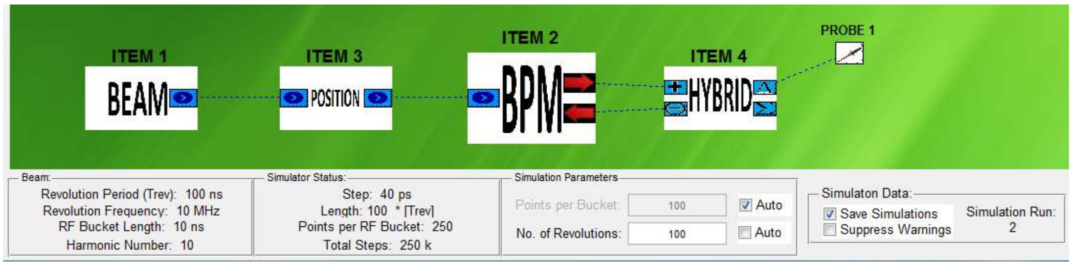
Simulation Parameters
Points per Bucket: 100 ☑ Auto
No. of Revolutions: 100 ☐ Auto

Use a beam with a 100MHz RF structure comprised of 2ns Gaussian bunches and simulate over 100 turns.

- ***a) What harmonic number needs to be chosen in the "BEAM" block to generate such a beam?***
- ***b) What do you notice about the output electrode signals?***

9) In order to calculate the horizontal position we need to obtain the difference between the left and right electrode amplitudes. This can be done using an electronic device called a hybrid. A hybrid can be an active or passive electronic device and simply has the role of adding and subtracting the input signals over a given frequency range. In the simulation the hybrid works over the whole frequency range and simply provides a sum (Σ) and difference (Δ) output of the two input signals.

*Add a **"HYBRID"** from the analogue blocks (Common Mode Rejection Ratio = 100%, i.e. the difference output is perfect with no leakage from the sum output) to the previous circuit as shown below, probing only the difference (Δ) output, and re-simulate.*



Beam:
Revolution Period (Trev): 100 ns
Revolution Frequency: 10 MHz
RF Bucket Length: 10 ns
Harmonic Number: 10

Simulator Status:
Step: 40 ps
Length: 100 * [Trev]
Points per RF Bucket: 250
Total Steps: 250 k

Simulation Parameters
Points per Bucket: 100 ☑ Auto
No. of Revolutions: 100 ☐ Auto

Simulaton Data:
☑ Save Simulations
☐ Suppress Warnings
Simulation Run: 2

Now increase the bunch intensity by a factor of two, re-simulate and compare to the signal from half the intensity.

- ***a) What do you notice about the difference signal?***

10) In order to use the difference signal to find a position it is necessary to NORMALISE the signal, i.e. to make it intensity independent. This is achieved by dividing the difference (Δ) output by the sum (Σ) output.

*Insert a division **"/"** block from the mathematical operations section, re-set the intensity to 5×10^9 and simulate. Double the bunch intensity, re-simulate and compare the output signal from the two different intensities.*

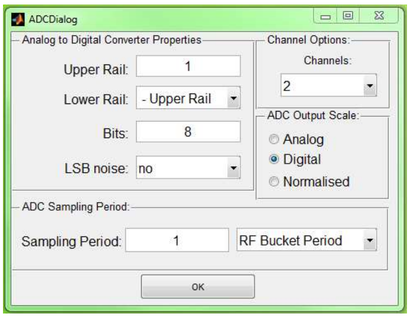
- ***a) What do you now notice about the difference signal?***
- ***b) Work out what position a Normalised output of 0.4 corresponds to?***

Beam Position Acquisition Systems:

1. Analogue to Digital Conversion:

Now that we know how to obtain an intensity independent position reading the next stage is to turn this into something that can be displayed in the control room. As it is difficult to perform the signal division block with analogue electronics the usual way of doing this is to digitise the difference (Δ) and sum (Σ) outputs from the hybrid and then perform the division in the digital domain. This can be done directly in a computer or using digital processing electronics such as a Field Programmable Gate Array (FPGA) or dedicated Digitial Signal Processing (DSP) chip.

The ***"Analogue to Digital Converter"*** (ADC) from the Special Blocks has the following properties:



Analog to Digital Converter Properties
Upper Rail: 1
Lower Rail: - Upper Rail
Bits: 8
LSB noise: no

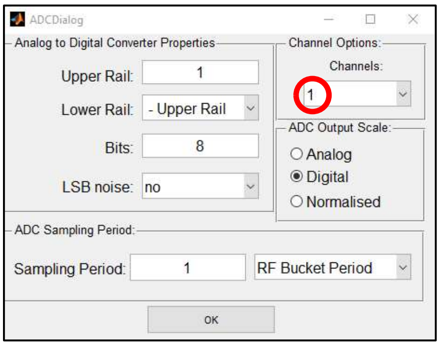
Channel Options:
Channels: 2

ADC Output Scale:
○ Analog
● Digital
○ Normalised

ADC Sampling Period:
Sampling Period: 1 RF Bucket Period

- The upper & lower rail voltages define the input range that the ADC can handle, in the example shown ±1V.
- The number of bits defines the granularity with which this input range is quantised, typically ranging from 8bits for systems sampling at >1GSamples/s to 24bits for systems sampling at <200kSamples/s. In the example shown an 8 bit ADC has been defined, which means that the ±1V input range is split into 2^8 (=256) digital levels.
- The number of bits of noise that the ADC has. This is typically a few bits, but in the tool is limited to the LEAST SIGNIFICANT BIT (LSB), and can be switched ON/OFF.
- The number of channels, which allows the same block to be used to digitise multiple inputs.
- The output scale
    - Analog in Volts
    - Digital in bits (from $-\frac{1}{2} \times 2^{\text{No. of bits}}$ to $+\frac{1}{2} \times 2^{\text{No. of bits}}$)
    - Normalised from -1 to 1
- The time at which the ADC samples the input waveform. In the example shown the waveform is sampled once every RF bucket.

11) *Add the following **"Analogue to Digital Converter"** (ADC) from the Special Blocks after the hybrid.*



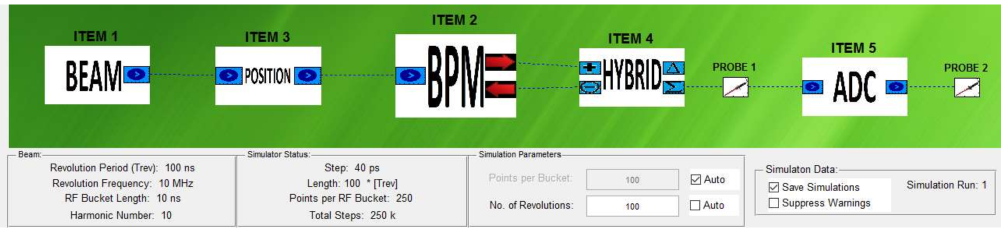
Analog to Digital Converter Properties:
Upper Rail: 1
Lower Rail: - Upper Rail
Bits: 8
LSB noise: no

Channel Options:
Channels: 1

ADC Output Scale:
○ Analog
● Digital
○ Normalised

ADC Sampling Period:
Sampling Period: 1 RF Bucket Period

Let's initially look at only the sum (Σ) output. Simulate the system for a bunch intensity of 5×10^9.



Beam:
Revolution Period (Trev): 100 ns
Revolution Frequency: 10 MHz
RF Bucket Length: 10 ns
Harmonic Number: 10

Simulator Status:
Step: 40 ps
Length: 100 * [Trev]
Points per RF Bucket: 250
Total Steps: 250 k

Simulation Parameters
Points per Bucket: 100 ☑ Auto
No. of Revolutions: 100 ☐ Auto

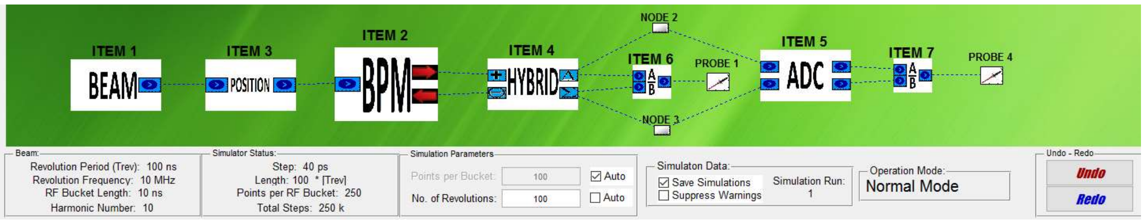
Simulaton Data:
☑ Save Simulations
☐ Suppress Warnings
Simulation Run: 1

a) ***To what voltage difference does the least significant bit of the ADC correspond (i.e. an increment of 1 when representing the ADC output in decimal)?***

b) ***At what bunch intensity does the circuit saturate for large position offsets?***

Now let's look at measuring position. To obtain the position we again need to normalise the signal to make it independent of intensity.

*Replace the one channel ADC with a 2 channel ADC digitising both the sum (Σ) and difference (Δ) signals after the hybrid. Insert a division **"/"** block from the mathematical operations section, this time using the digitised data at the output of the ADC. You can also insert another **"/"** block just after the hybrid to allow a comparison of the analogue and digital output data.*



Beam:
Revolution Period (Trev): 100 ns
Revolution Frequency: 10 MHz
RF Bucket Length: 10 ns
Harmonic Number: 10

Simulator Status:
Step: 40 ps
Length: 100 * [Trev]
Points per RF Bucket: 250
Total Steps: 250 k

Simulation Parameters
Points per Bucket: 100 ☑ Auto
No. of Revolutions: 100 ☐ Auto

Simulaton Data:
☑ Save Simulations
☐ Suppress Warnings
Simulation Run: 1

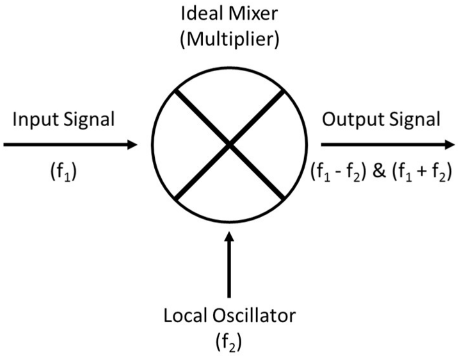
Operation Mode:
Normal Mode

c) ***Looking at the difference signal at the centre of the beampipe (where the sum signal remains constant) determine the maximum resolution of this system.***

d) ***How many bits do we need to have a resolution of 190 micrometres for a maximum bunch intensity of* 5×10^9 ?**

2. Homodyne and Heterodyne receivers

Heterodyning is a radio signal processing technique invented in 1901 by Canadian inventor-engineer Reginald Fessenden that creates new frequencies by combining or mixing two frequencies. Heterodyning is used to shift one frequency range into another, new one, and is also involved in the processes of modulation and demodulation. The two frequencies are combined in a nonlinear signal-processing device such as a vacuum tube, transistor, or diode, usually called a mixer.



In the most common application, two signals at frequencies f1 and f2 are mixed, creating two new signals, one at the sum f1 + f2 of the two frequencies, and the other at the difference f1 − f2. These new frequencies are called heterodynes. Typically only one of the new frequencies is desired, and the other signal is filtered out of the output of the mixer.

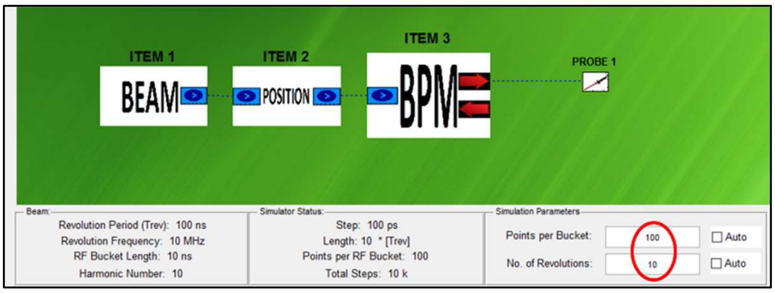
In a homodyne the input frequency (f_1) is itself used to generate the local oscillator frequency (f_2). The resulting frequencies become DC and $2f_1$, where the second harmonic is again filtered out, to leave only the DC component. In this way, a frequency component is converted into a DC signal level that is directly proportional to the original amplitude of the signal.

Such mixing techniques are often employed in beam position acquisition systems to move the frequency range of interest (i.e. the signal frequency) from that determined by the BEAM/BPM combination (often high or very high frequency) to a lower frequency which is easier to handle digitally, with lower frequency ADCs having more bits, and therefore better resolution.

Homodyne Circuits for Continuous Signals

Let’s look at how to construct a homodyne detection system for a circulating (continuous) beam.

12) *Construct the circuit as shown below:*



Beam:
Revolution Period (Trev): 100 ns
Revolution Frequency: 10 MHz
RF Bucket Length: 10 ns
Harmonic Number: 10

Simulator Status:
Step: 100 ps
Length: 10 * [Trev]
Points per RF Bucket: 100
Total Steps: 10 k

Simulation Parameters
Points per Bucket: 100 ☐ Auto
No. of Revolutions: 10 ☐ Auto

- *The **“BEAM”** should have the following parameters:*
  - *i. Revolution period 100ns*
  - *ii. Harmonic number 10*
  - *iii. Intensity 1×10^{12} charges*
  - *iv. Cos^2 distribution*
  - *v. Bunch length 10ns with 1 bunch in every RF bucket*
- *The **“POSITION”** should be set to “CONSTANT” and equal to zero*
- *The **“BPM**” can be the same BPM constructed in Q7 of Part 1*

Run the simulation and observe the frequency and time domain signals.

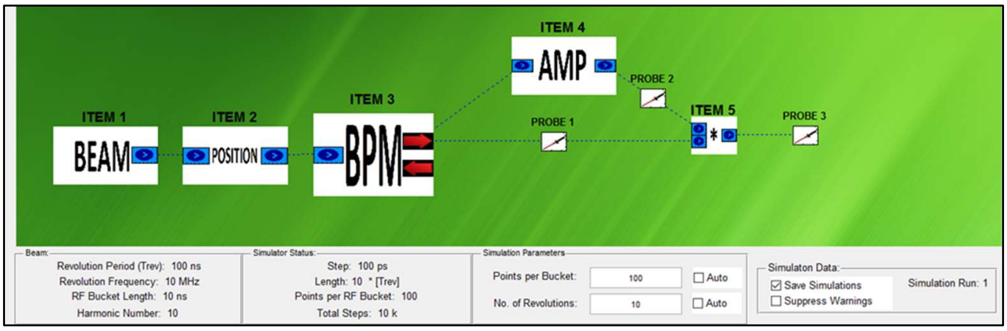
a) What do you observe about the frequency content for this particular “BEAM”?

To create a homodyne circuit we need to generate a local oscillator (f_2) that is an intensity independent signal (i.e. constant amplitude signal) with the same frequency as the input signal (f_1). This can be done using a limiting amplifier to produce a square wave from the input signal.

Add an amplifier stage and try to obtain such a signal.

b) What are the parameters of your amplifier?

Now mix the two signals together using a multiplication “” block from the mathematical operations section. This will represent our homodyne mixer.*



Beam:
Revolution Period (Trev): 100 ns
Revolution Frequency: 10 MHz
RF Bucket Length: 10 ns
Harmonic Number: 10

Simulator Status:
Step: 100 ps
Length: 10 * [Trev]
Points per RF Bucket: 100
Total Steps: 10 k

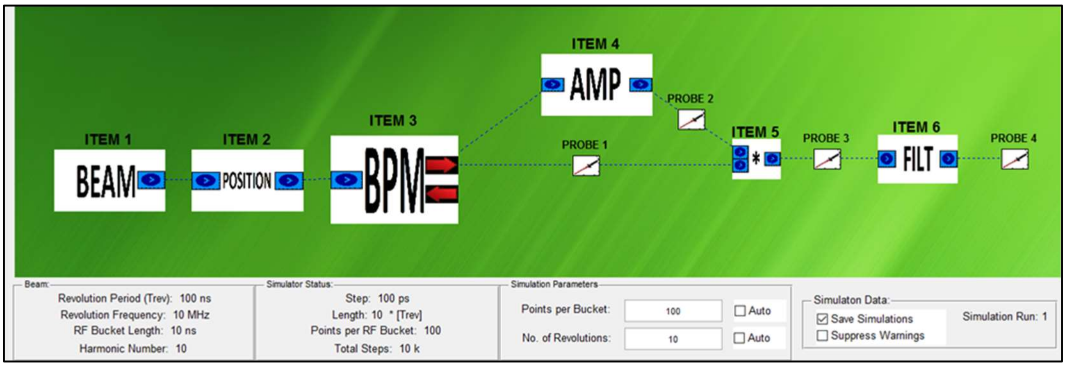
Simulation Parameters
Points per Bucket: 100 ☐ Auto
No. of Revolutions: 10 ☐ Auto

Simulaton Data:
☑ Save Simulations
☐ Suppress Warnings
Simulation Run: 1

Run the simulation again and observe the frequency and time domain signals (zoom in between 0-100ns for the time domain).

c) What do you observe about the frequency content before and after the mixer (multiplier)?

In the end, we only want to retain the DC signal. Insert a filter block and adjust the parameters such that a steady state output signal is obtained by the mid-point of the simulation (i.e. 500ns).



Beam:
Revolution Period (Trev): 100 ns
Revolution Frequency: 10 MHz
RF Bucket Length: 10 ns
Harmonic Number: 10

Simulator Status:
Step: 100 ps
Length: 10 * [Trev]
Points per RF Bucket: 100
Total Steps: 10 k

Simulation Parameters
Points per Bucket: 100 ☐ Auto
No. of Revolutions: 10 ☐ Auto

Simulaton Data:
☑ Save Simulations
☐ Suppress Warnings
Simulation Run: 1

d) What does the response time of the filter depend on?

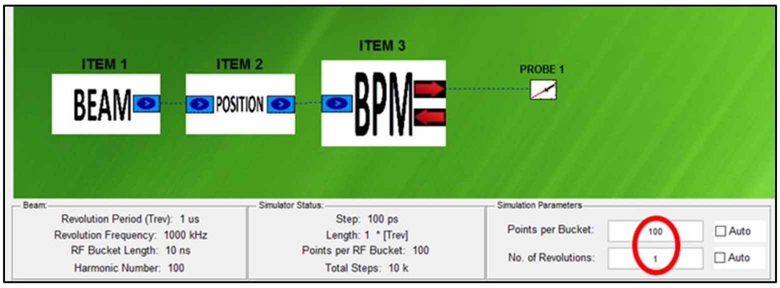
e) What is the ratio of your DC component to the second revolution harmonic?

f) How can this be further increased without impacting the overall response time?

Homodyne Circuits for Pulsed Signals

Homodyne circuits, such as the one above, can also be used for pulsed signals as long as the pulse length allows a steady state signal to be reached (e.g. measurement of the position in a linac or transfer line). When the pulse is very short, other tactics have to be employed. In this case the trick is to use the beam to excite a ***bandpass filter*** that rings for long enough for the homodyne circuitry to work. Let's now look at how to construct a homodyne detection system for a short, single pass bunch.

13) *Construct the circuit as shown below:*



Beam:
Revolution Period (Trev): 1 us
Revolution Frequency: 1000 kHz
RF Bucket Length: 10 ns
Harmonic Number: 100

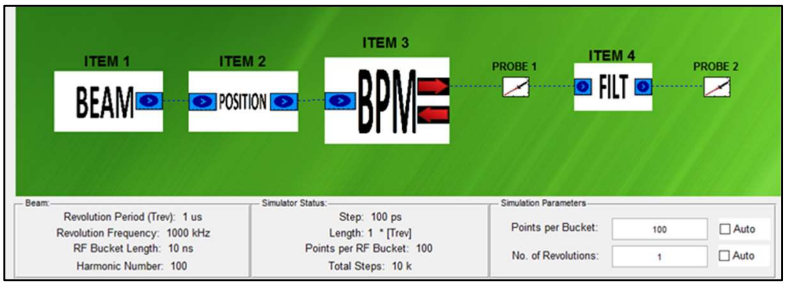
Simulator Status:
Step: 100 ps
Length: 1 * [Trev]
Points per RF Bucket: 100
Total Steps: 10 k

Simulation Parameters
Points per Bucket: 100 ☐ Auto
No. of Revolutions: 1 ☐ Auto

- *The **"BEAM"** should have the following parameters (and note that the "No. of Revolutions:" on the main tab should be set to 1:*
    - *i. Revolution period 1000ns*
    - *ii. Harmonic number 100*
    - *iii. Intensity 1×10^{12} charges*
    - *iv. Cos^2 distribution*
    - *v. Bunch length 10ns with 1 bunch every 100 RF bucket*

  This simulates a single passage for a single bunch over the same time period (1μs) as in the continuous case
- *The **"POSITION"** should be set to "CONSTANT" and equal to zero*
- *The **"BPM"** can be the same BPM constructed in Q7 of Part 1*

*Now add a filter block after the BPM signal. Try to find **bandpass filter** settings that result in the highest amplitude for a ring time that is at least half the simulation time of 1μs.*



Beam:
Revolution Period (Trev): 1 us
Revolution Frequency: 1000 kHz
RF Bucket Length: 10 ns
Harmonic Number: 100

Simulator Status:
Step: 100 ps
Length: 1 * [Trev]
Points per RF Bucket: 100
Total Steps: 10 k

Simulation Parameters
Points per Bucket: 100 ☐ Auto
No. of Revolutions: 1 ☐ Auto

a) What do you notice about the relationship between frequency, amplitude and ring time?

Adapt the signals such that you can re-use the homodyne stage developed for the continuous beam to also process the pulsed beam signal.

b) What is the important element to add?

14) As a final task, select one of the two homodyne systems investigated and build a complete beam position system based on this technique with the following specifications:

- *Dynamic range: bunch intensity from* 1×10^{10} *to* 1×10^{12} *charges*
- *Resolution: better than* $150\mu m$ *for a centred beam*