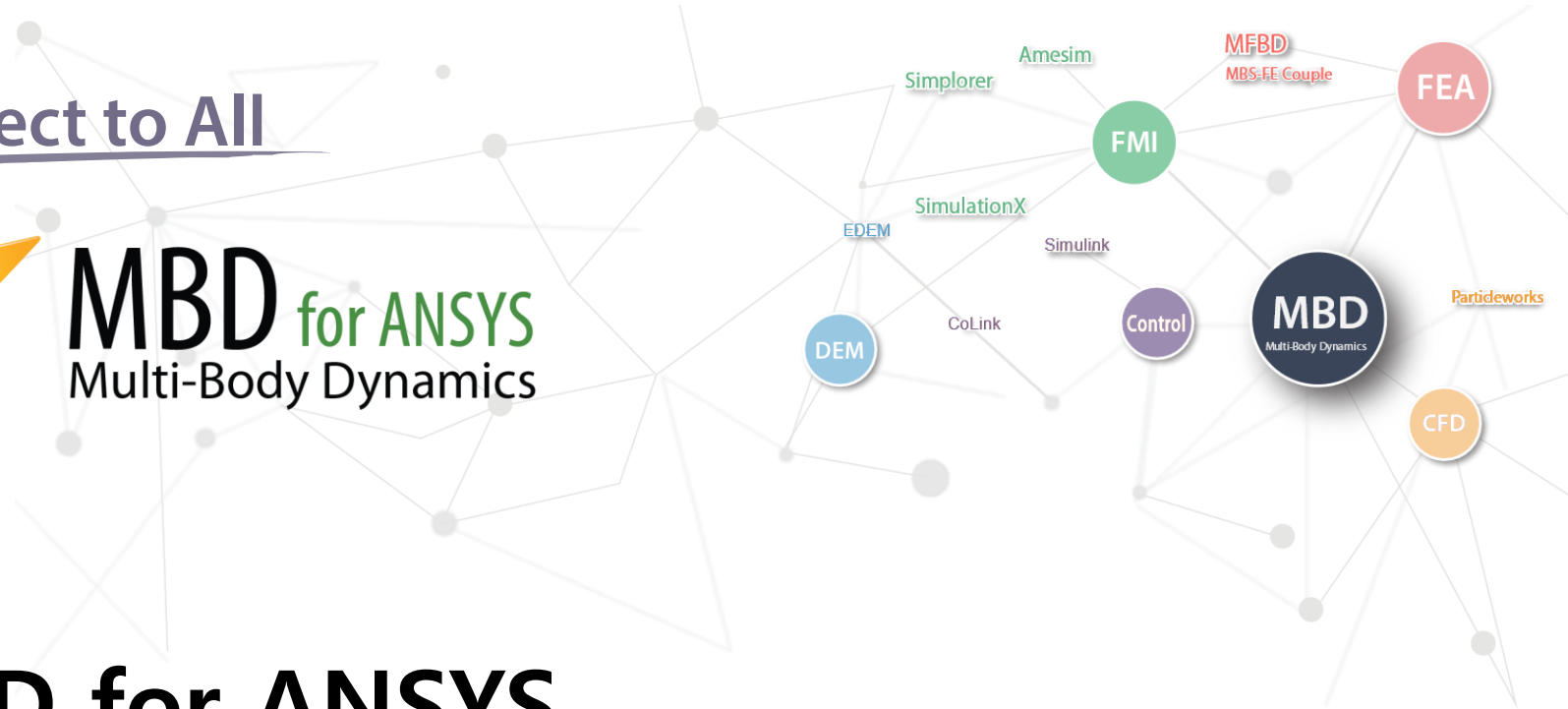


Connect to All



MBD for ANSYS
Multi-Body Dynamics



MBD for ANSYS Simulink Interface Training

James Kang

Korean

English

FunctionBay, Inc. Extended Application Team
Senior Manager (mbd4a@functionbay.co.kr)

Index

❑ MBD for ANSYS Model

1. Create a new system
2. Importing Geometry
3. Unit and Gravity
4. Delete Contact Regions
5. Joint
6. Simulation
7. Plant Input & Plant Output
8. Analysis setting
9. Create Simulink Interface

❑ Matlab Model

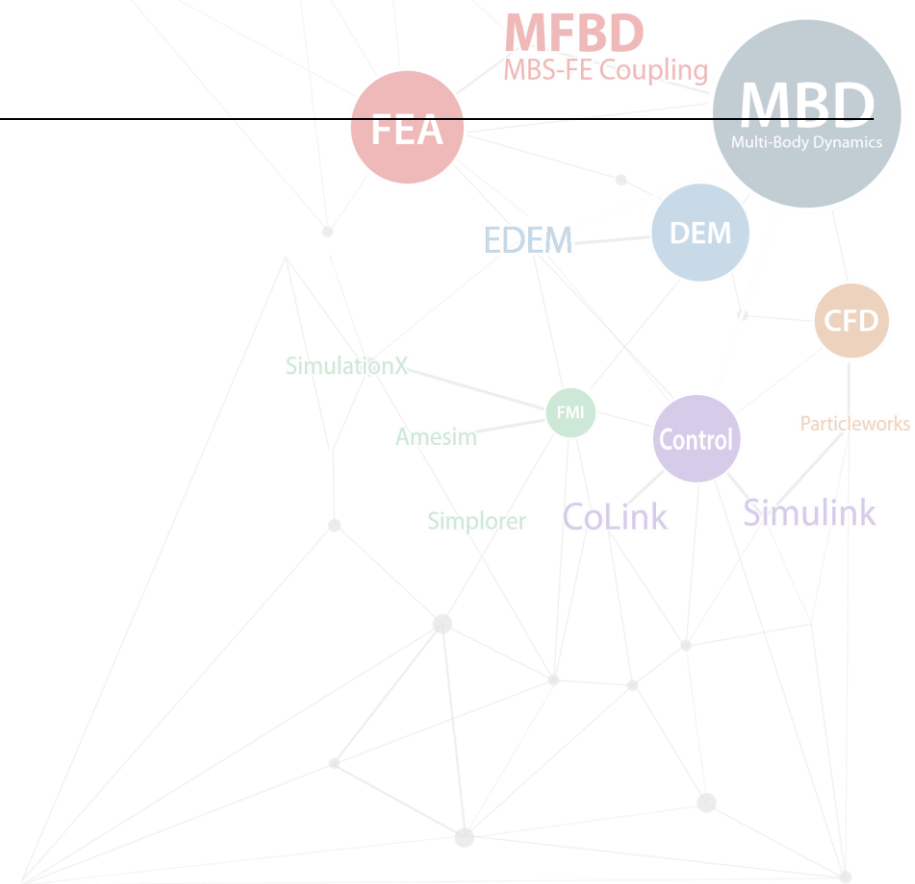
1. Current Directory
2. MBD for ANSYS Host Block
3. Simulink model

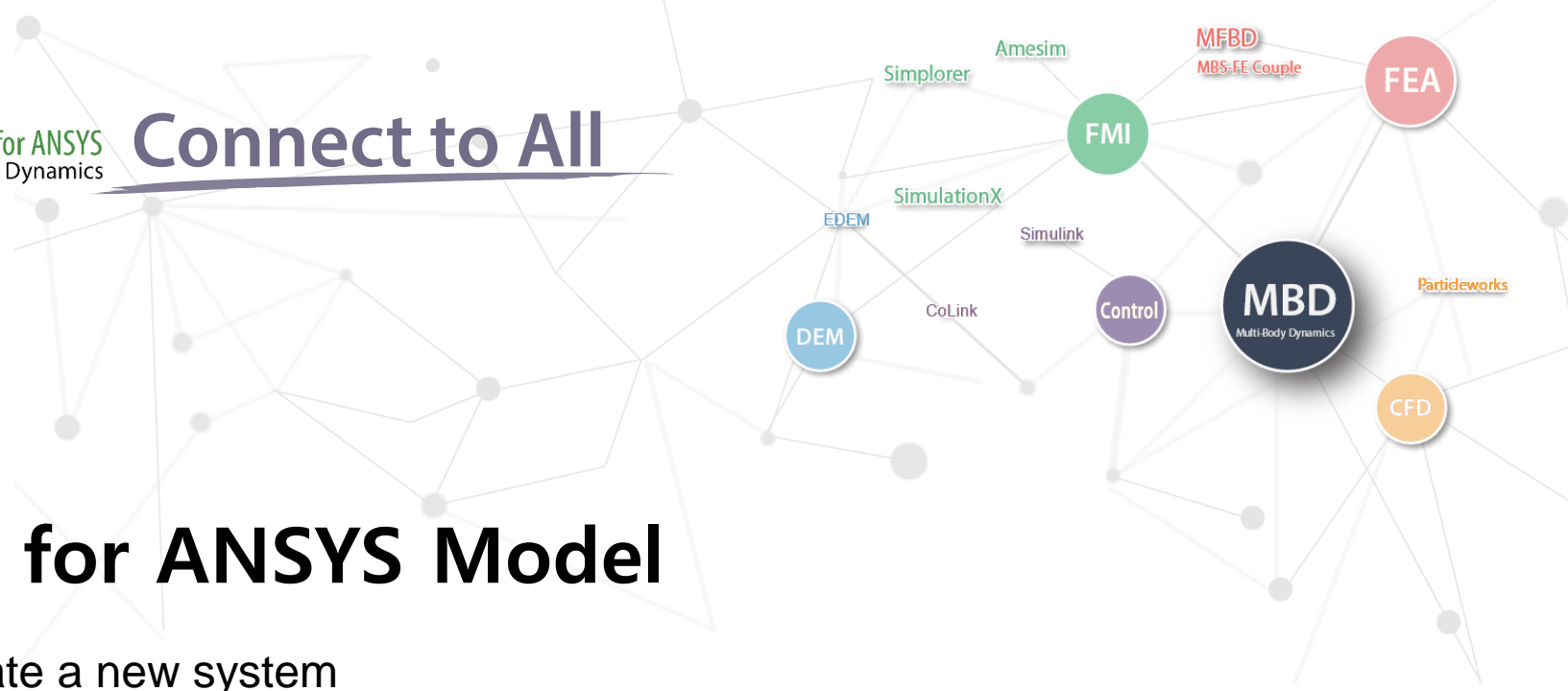
❑ Simulation

1. Simulation
2. Post (Animation, Plot)

❑ Improve Simulink Model

1. Adding PID control block
2. Post (Animation, Plot)





MBD for ANSYS Model

1. Create a new system
2. Importing Geometry
3. Unit and Gravity
4. Delete Contact Regions
5. Joint
6. Simulation
7. Plant Input & Plant Output
8. Analysis setting
9. Create Simulink Interface

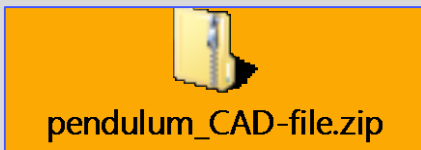
Create a new system

1. Create a Multi-Body Dynamics system in the Project Schematic window
2. **Right mouse button** with the cursor over the **Geometry** field
3. Select **Import Geometry – Browse**
4. **Select pendulum.x_t** file and import it
5. **Right mouse button** with the cursor over the **Geometry** field and select the **Edit Geometry in DesignModeler**

The image shows a sequence of four screenshots illustrating the process of creating a new system in ANSYS Workbench:

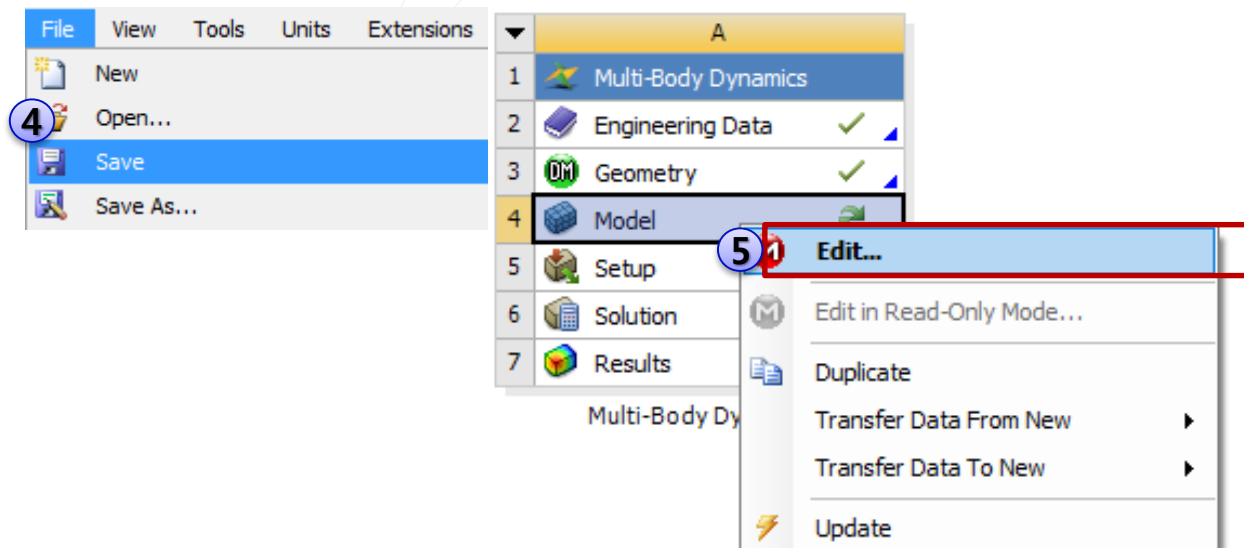
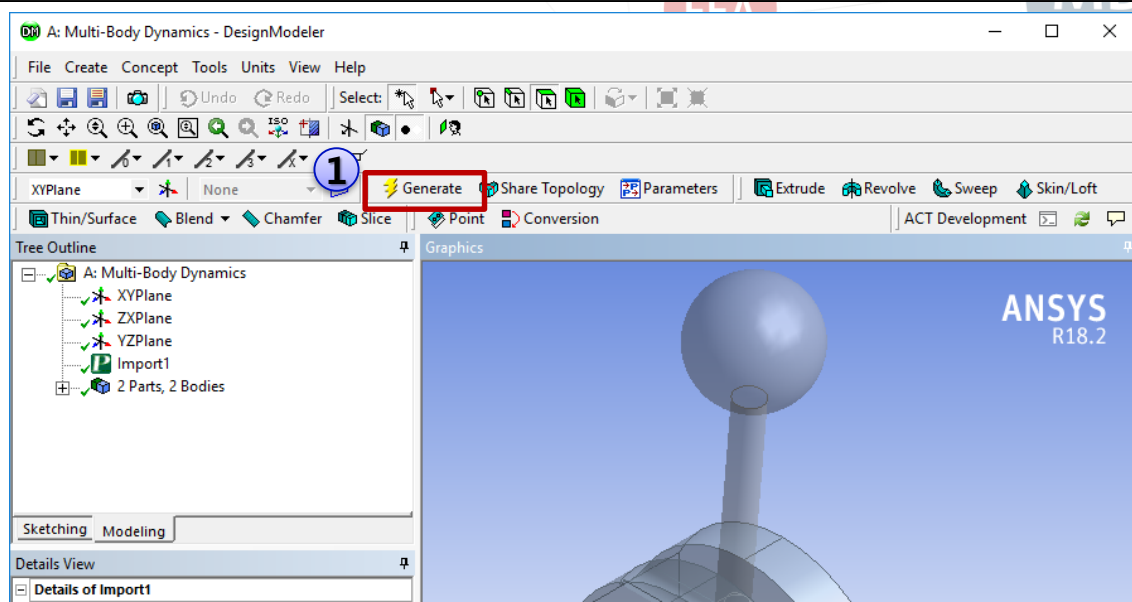
- Step 1:** The Project Schematic window shows the Toolbox on the left. The 'Multi-Body Dynamics' option is highlighted with a red box and a circled '1'.
- Step 2:** The Project Schematic window shows the 'Geometry' field selected with a right mouse button click. A red box highlights the 'Geometry' field, and a callout box with a red arrow points to it with the text '마우스 우 클릭' (Right mouse click).
- Step 3:** The context menu for the 'Geometry' field is open. The 'Import Geometry' option is selected, and the 'Browse...' sub-option is highlighted with a red box and a circled '3'.
- Step 4:** The context menu is open, and the 'Edit Geometry in DesignModeler...' option is highlighted with a red box and a circled '5'. A file named 'pendulum.x_t' is visible in the background.

A network diagram on the right side of the image shows various simulation methods: FEA, MBD, DEM, CFD, Particleworks, CoLink, Simulink, and MI, all interconnected.



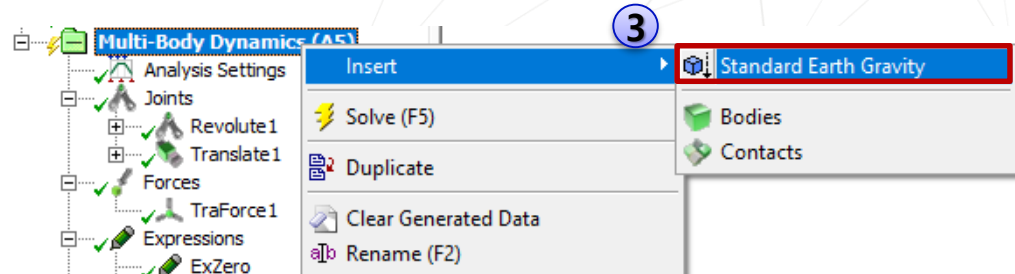
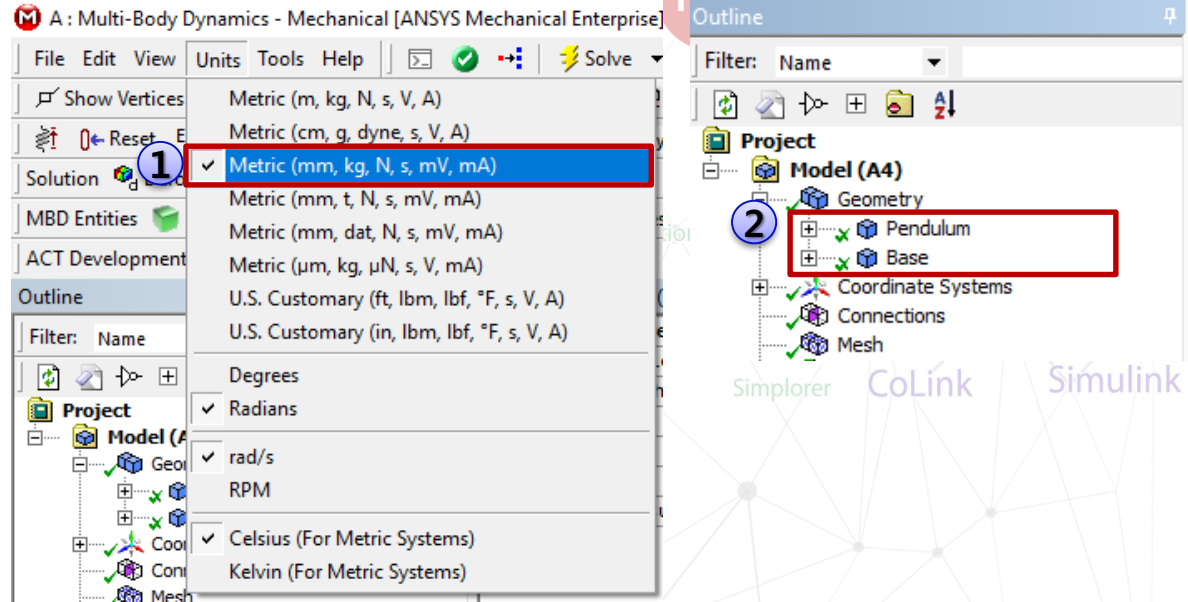
Importing Geometry

1. Click **Generate** tool in the toolbar of DesignModeler.
2. You can see the geometry.
3. **Close** DesignModeler
4. **Save** the project
5. **Right mouse button** with the cursor over the **Model** field and select the **Edit...**



Unit and Gravity

1. Set the **units** to **Metric (mm, kg, N, s, mV, mA)**
2. Rename the geometries to **Pendulum** and **Base**
3. Right mouse button with the cursor over the Multi-Body Dynamics in the outline window and select: **Insert – Standard Earth Gravity**
4. Change the **Direction** to **-Y Direction**

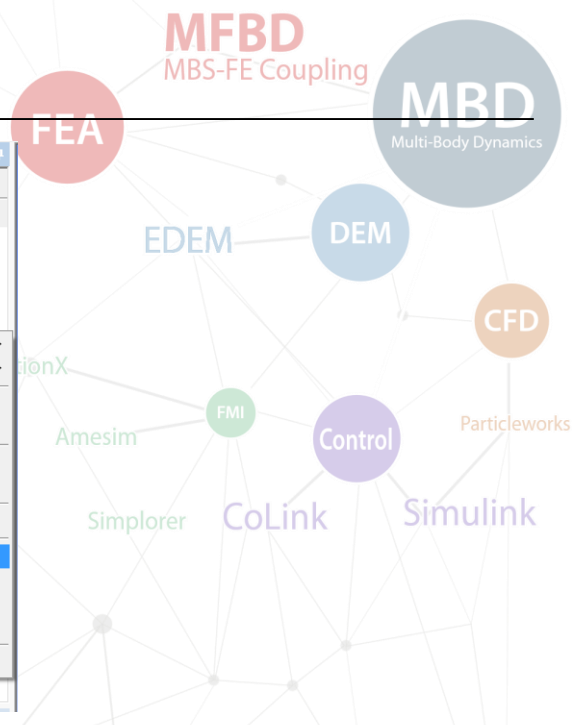
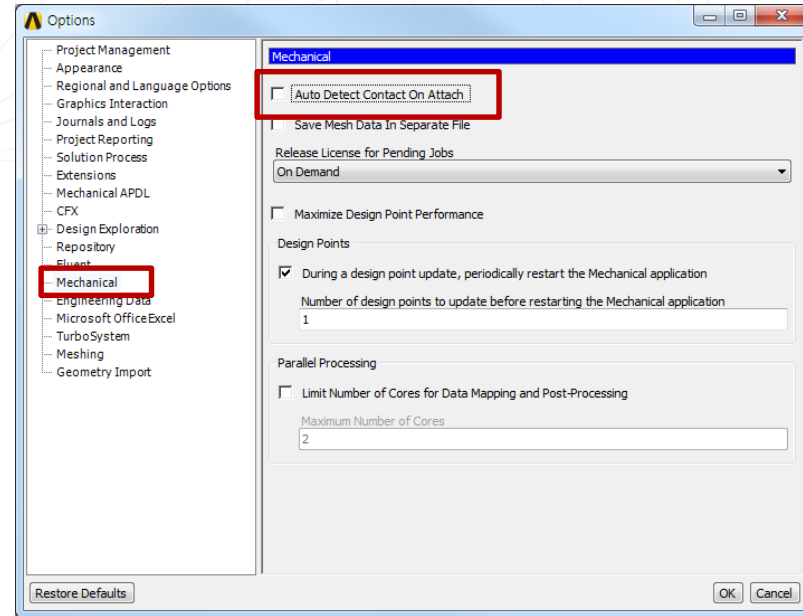
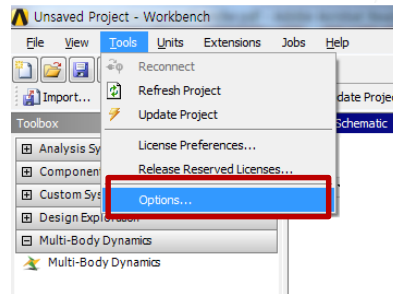
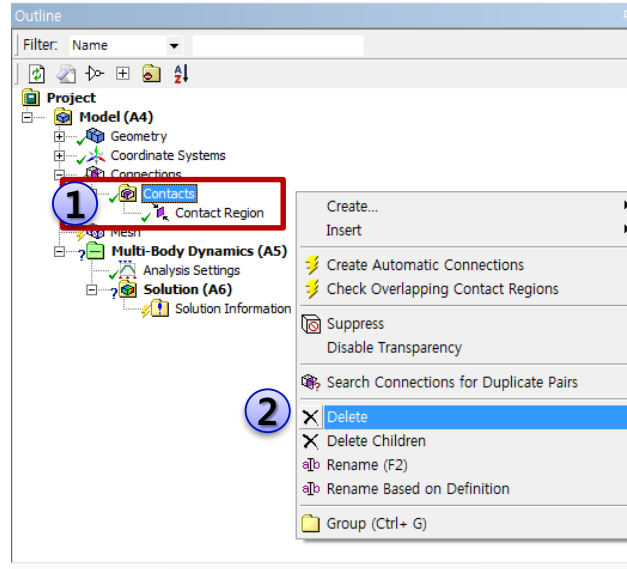


Details of "Standard Earth Gravity"

Scope	
Geometry	All Bodies
Definition	
Coordinate System	Global Coordinate System
X Component	0. mm/s ² (ramped)
Y Component	-9806.6 mm/s ² (ramped)
Z Component	0. mm/s ² (ramped)
Suppressed	No
Direction	-Y Direction

Delete Contact Region

1. Select Contacts in the Outline window.
2. Select **Delete** from the popup menu.
3. You should **uncheck** the 'Auto Detect Contact on Attach' option for **Mechanical** if you don't want to repeat this 'Delete' operation later. (**Tools – Options... in the Workbench Window**)



Joint – Revolute Joint

1. In the MBD Entities Toolbar, select **Joint – Revolute**
2. Set **Connection Type** to **Body-Body**
3. Select '**Base**' for the Base Body
4. Select '**Pendulum**' for the Action Body
5. Set Origin to **0,0,0**
6. Set Rotational Axis to **0,0,1**

MBD Entities **Joints** Forces Contacts Expressions

1 Fixed
2 **Revolute**
3 Translate
4 Planar
5 Cylindrical
6 Spherical
Screw
Universal
Con.Veloc
CMotion
PointCurve
CurveCurve
Coupler
Gear

Details of "Revolute1"

Definition	
Connection Type	Body-Body
Coordinate System	No
Base Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	Base
Action Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	Pendulum
Origin	
Origin	Click to Change
<input type="checkbox"/> Origin X	0 [mm]
<input type="checkbox"/> Origin Y	0 [mm]
<input type="checkbox"/> Origin Z	0 [mm]
Rotational Axis	
Axis	Click to Change
<input type="checkbox"/> Axis X	0
<input type="checkbox"/> Axis Y	0
<input type="checkbox"/> Axis Z	1
Motion	
Include Motion	No
Friction	
Include Friction	No

A: Multi-Body Dynamics
Revolute1
2018-06-20 12:57 PM
 Revolute1

Joint – Translational Joint

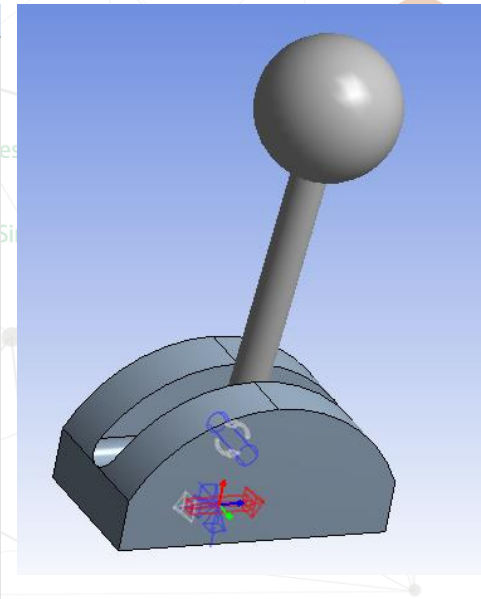
1. In the MBD Entities Toolbar, select **Joint – Translate**
2. Set **Connection Type** to **Ground-Body**
3. Select '**Base**' for the Action Body
4. Set Origin to **0,-100,0**
5. Set Rotational Axis **1,0,0**

MBD Entities **Joints** Forces Contacts Expressions

Fixed
Revolute
Translate
Planar
Cylindrical
Spherical
Screw
Universal
Con.Veloc
CMotion
PointCurve
CurveCurve
Coupler
Gear

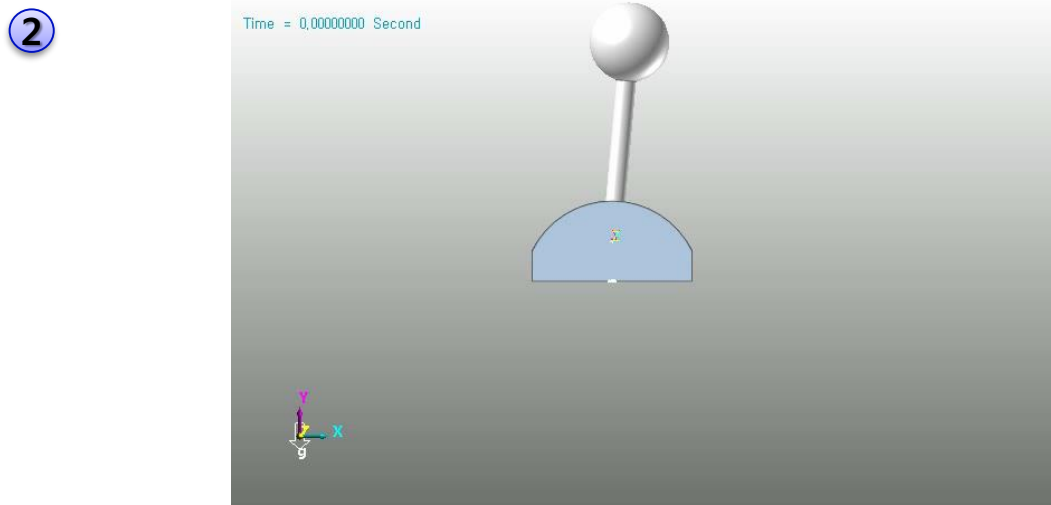
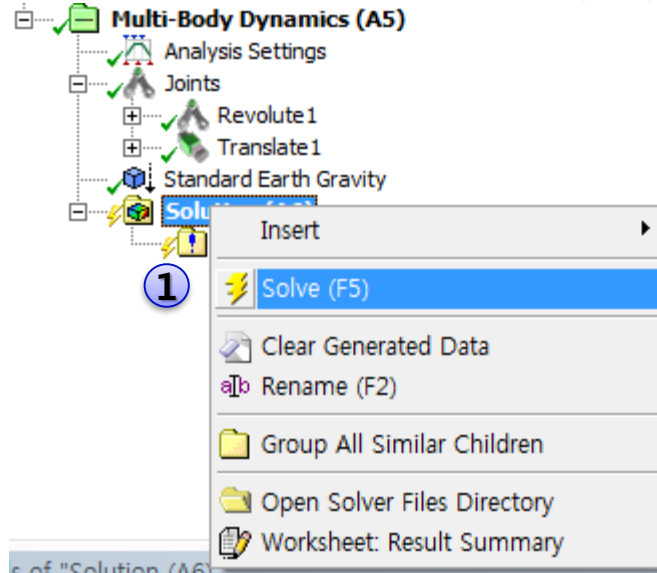
Details of "Translate1"

Definition	
Connection Type	Ground-Body
Coordinate System	No
Base Body Information	
Body	Ground
Action Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	Base
Origin	
Origin	Click to Change
<input type="checkbox"/> Origin X	0 [mm]
<input type="checkbox"/> Origin Y	-100 [mm]
<input type="checkbox"/> Origin Z	0 [mm]
Translational Axis	
Axis	Click to Change
<input type="checkbox"/> Axis X	1
<input type="checkbox"/> Axis Y	0
<input type="checkbox"/> Axis Z	0
Motion	
Include Motion	No
Friction	
Include Friction	No

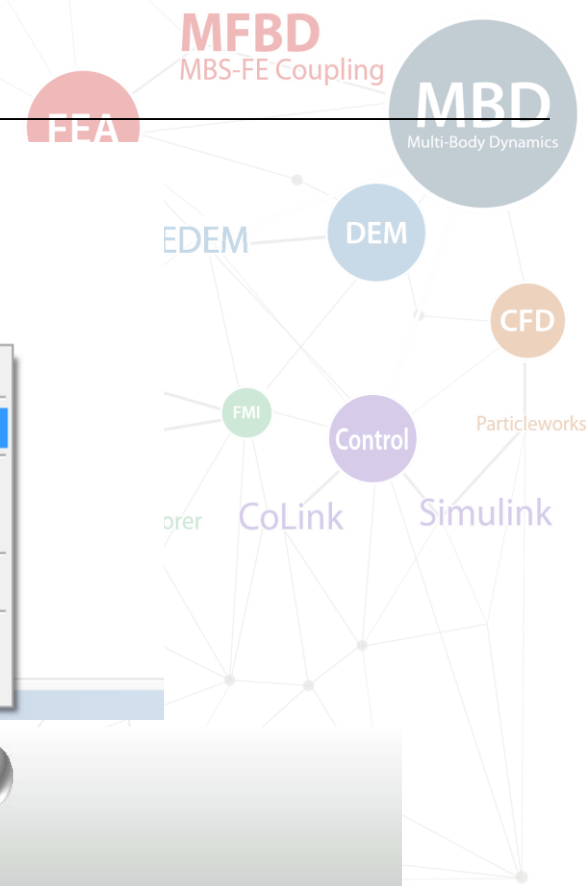


Simulation

1. **Solve** the model
2. You can see the **animation**
3. You will add the PID controller for the pendulum in the next pages.

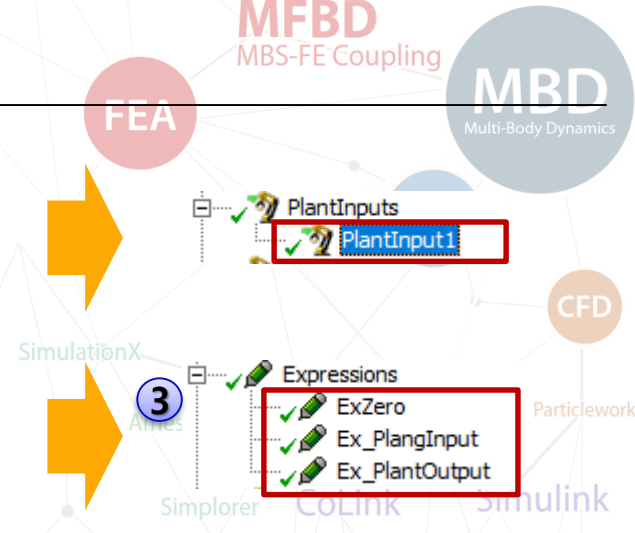
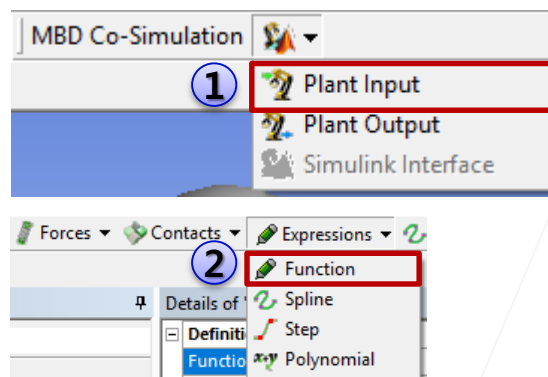


Play the animation

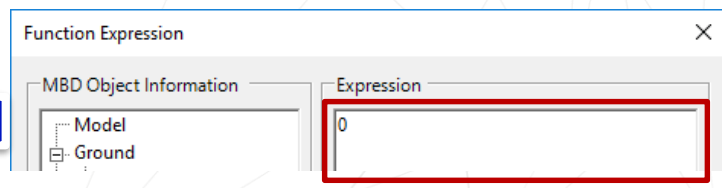


Plant Input & Plant Output

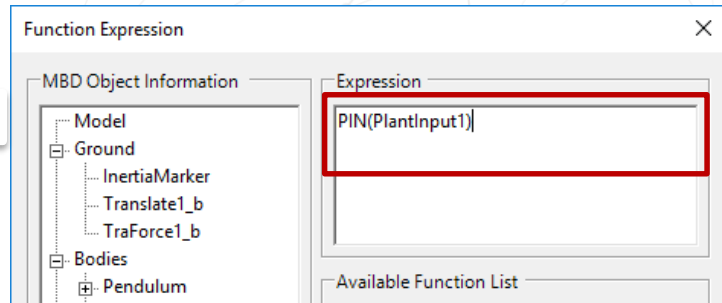
1. In the MBD Co-Simulation Toolbar, select **Plant Input**
2. In the MBD Entities Toolbar, select **Expressions – Function**
3. **Rename** the name of the function Expression to **ExZero** and **create 2 more function expressions** and **rename** them to **Ex_PlantInput**, **Ex_PlantOutput**
4. Input **0** for ExZero
5. Input **PIN(PlantInput1)** for Ex_PlantInput
6. Input **(AZ(Pendulum.Revolute1_a, Base.Revolute1_b)-5D)** for Ex_PlantOutput



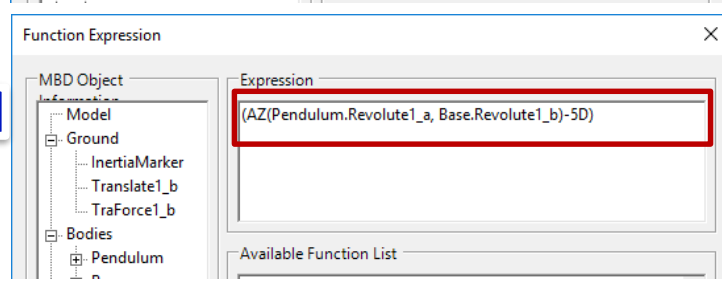
4
ExZero



5
Ex_PlantInput

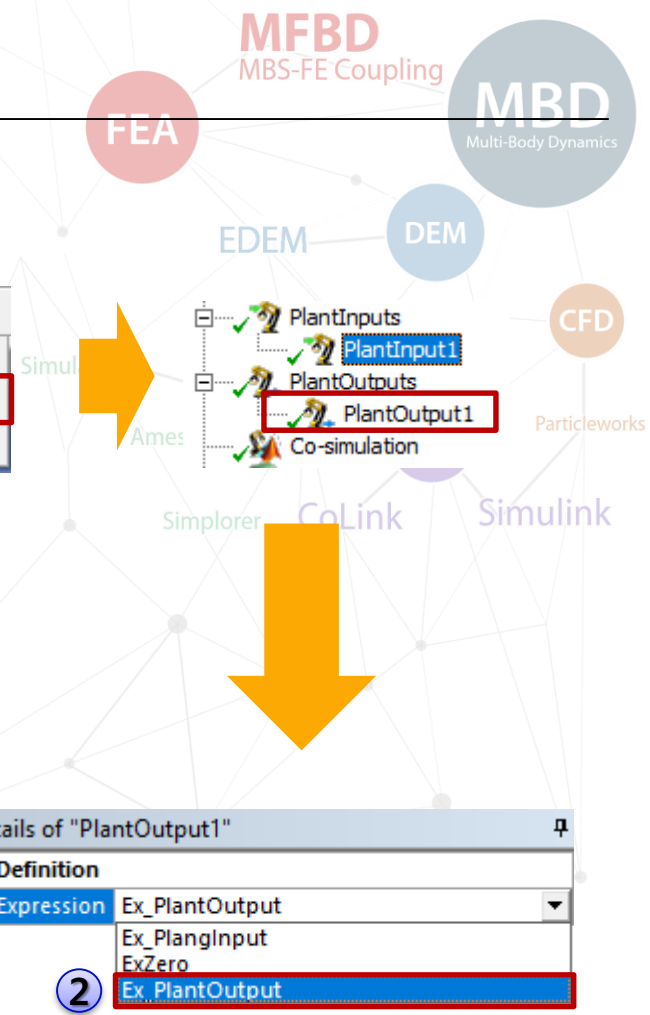
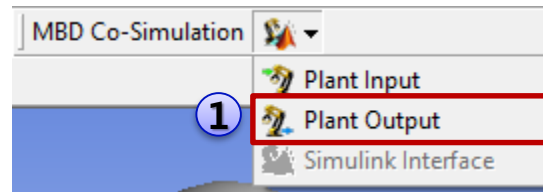


6
Ex_PlantOutput



Plant Input & Plant Output

1. In the MBD Co-Simulation Toolbar, select **Plant Output**
2. Select **Ex_PlantOutput** for the expression



Force – Translational Force

1. In the MBD Entities Toolbar, select Force – **Tra.Force**
2. Set **Connection Type** to **Ground-Body**
3. Select '**Base**' for the Base Body
4. Set Origin **0,-100,0**
5. Select **Ex_PlantInput** for **FX Function Exp**, and select **ExZero** for **FY Function Exp** and **FZ Function Exp**

The screenshot shows the MBD Entities Toolbar with 'Forces' selected. The 'Details of "TraForce1"' dialog box is open, showing the following configuration:

Definition	
Connection Type	Ground-Body
Coordinate System	No
Base Body Information	
Body	Ground
Action Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	Base
Origin	
Origin	Click to Change
<input type="checkbox"/> Origin X	0 [mm]
<input type="checkbox"/> Origin Y	-100 [mm]
<input type="checkbox"/> Origin Z	0 [mm]
Force	
FX Function Exp	Ex_PlantInput
FY Function Exp	ExZero
FZ Function Exp	ExZero
Reference Marker	
Use Reference Marker	No

Analysis Settings

1. Set the parameters of Analysis Settings
EndTime = 5
Step = 200
Maximum Time Step = 0.0001

Outline

Filter: Name

Project

- Model (A4)
 - Geometry
 - Pendulum
 - Base
 - Coordinate Systems
 - Connections
 - Mesh
 - Multi-Body Dynamics (A5)**
 - Analysis Settings**
 - Joints
 - Revolute 1
 - Translate 1
 - Forces
 - TraForce 1
 - Expressions
 - ExZero
 - Ex_PlantInput

Details of "Analysis Settings"

Definition

Simulation Type	Dynamics
-----------------	----------

General

<input type="checkbox"/> End Time	5 [sec]
<input type="checkbox"/> Step	200

Parameter

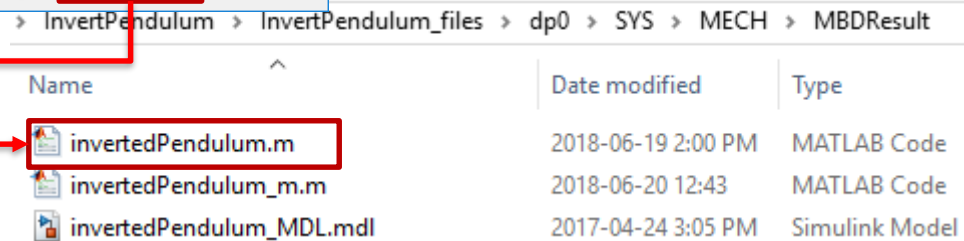
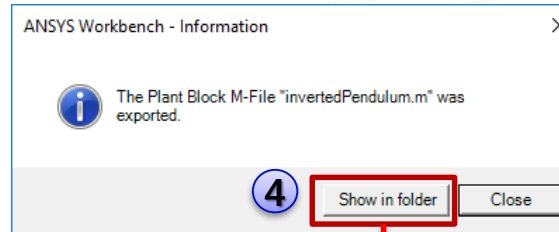
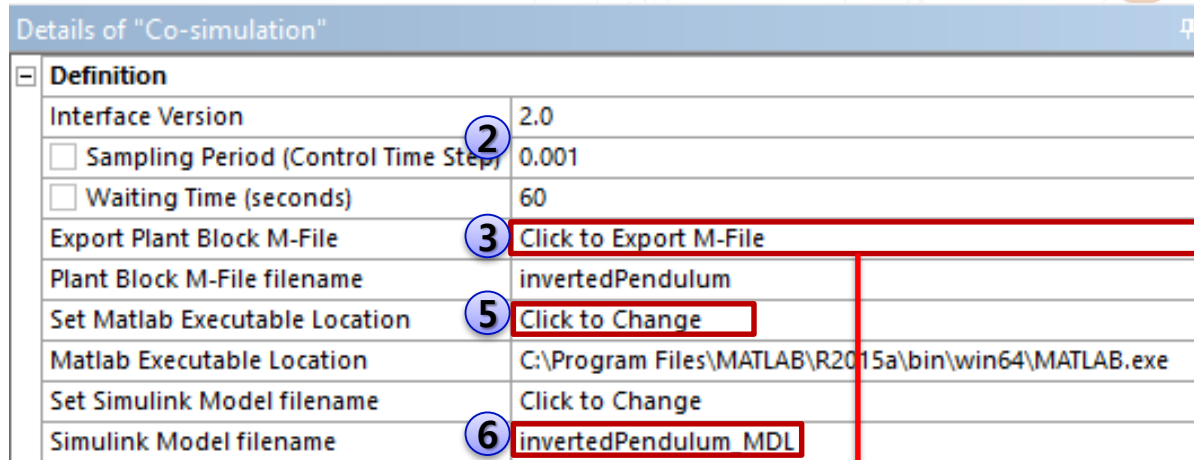
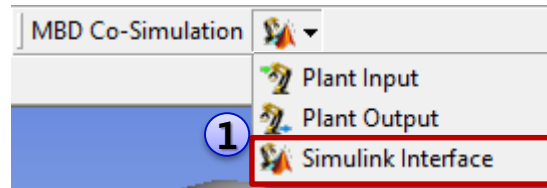
Maximum Order	2
<input type="checkbox"/> Maximum Time Step	0.0001
<input type="checkbox"/> Initial Time Step	1E-06
<input type="checkbox"/> Error Tolerance	0.005
Integrator Type	IMGALPHA
Numerical Damping	1
Constant Stepsize	1E-05
Jacobian Evaluation	100
Stop Condition	No

Include

Static Analysis	No
-----------------	----

Create Simulink Interface

1. In the MBD Co-Simulation Toolbar, select **Simulink Interface**
2. Set as below:
Interface Version = **2.0**
Sampling Period = **0.001**
(Set Interface Version = 3.0, if you use MATLAB 2016a or later)
3. Set Plant Block M-File filename to **invertedPendulum** and select '**Click to Export M-File**'
4. Click Show in folder and confirm if **invertedPendulum.m** is generated (in MBD result folder)
5. Select **Click to Change** and set the Matlab.exe for Matlab Executable Location (\$Matlabroot\bin\win**\MATLAB.exe)
6. Input Simulink Model filename to **invertedPendulum_MDL** (in MBD result folder) → This will be made in the next pages.





Matlab Model

1. Current Directory
2. MBD for ANSYS Host Block
3. Simulink model

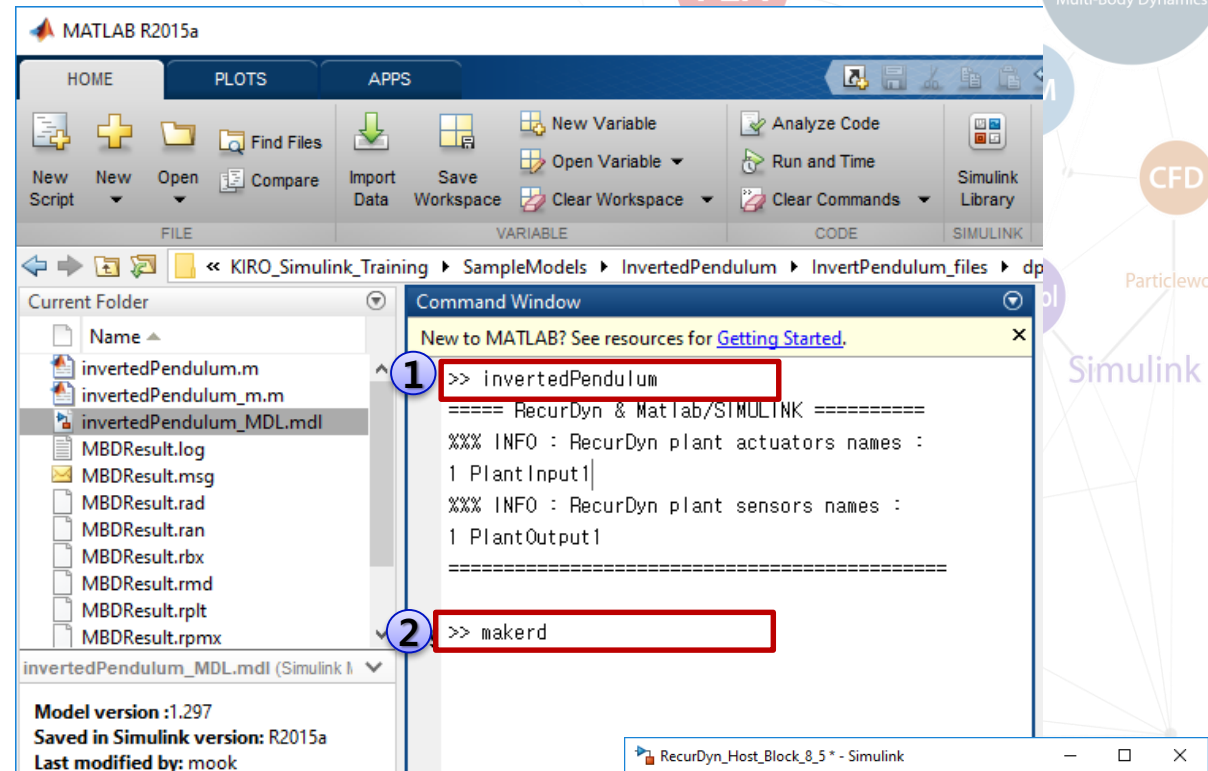
Working directory

1. Select **Open Solver Files Directory**
2. Select **MBDResult** folder
3. **Copy** the full path where **invertedPendulum.m** is located
4. Execute **Matlab** and **paste** the path to **Current Folder** of Matlab.

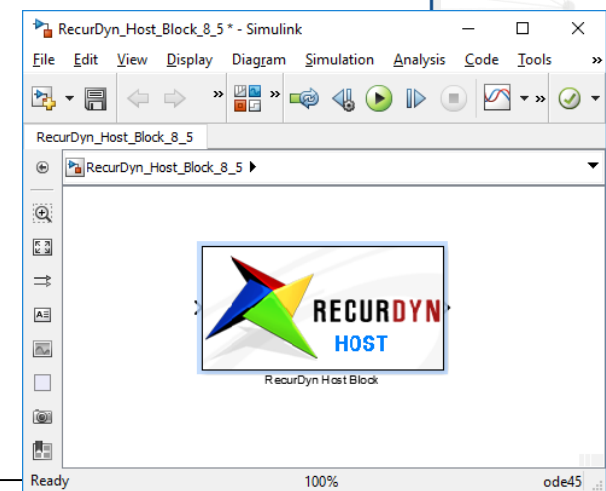
The image shows a sequence of steps to set the working directory in MATLAB. It starts with a file explorer window where a context menu is open over a folder structure. The 'Open Solver Files Directory' option is highlighted with a red box and a circled '1'. An arrow points to the 'MBDResult' folder, which is also highlighted with a red box and a circled '2'. Another arrow points to the address bar, which contains the full path: 'C:\Users\Training_Material\KIRO_Simulink_Training\SampleModels\InvertedPendulum\InvertPendulum_files\dp0\SYS\MECH\MBDResult'. A circled '3' is next to this path. A final arrow points to the MATLAB R2015a interface. In the 'Current Folder' pane, the same path is pasted and highlighted with a red box and a circled '4'. The Command window shows the MATLAB prompt 'fx >>'.

MBD for ANSYS Host Block

1. Input **invertedPendulum** in Command Window + [Enter]
2. Input **makerd** in the Command Window + [Enter]
3. **RecurDyn Host Block** is created



3
RecurDyn Host Block is created by makerd command



Simulink Model - Simulink Block

1. Select **Tools – Library Browser** in Simulink Window
2. Search **gain** in **Simulink Library Browser**
3. **Drag and Drop a Gain Block** to Simulink window where **RecurDyn Host Block** is.
4. Connect the blocks and set Gain to **-1800**

The image displays a sequence of four screenshots from the Simulink software interface, illustrating the process of adding a Gain block to a model.

Step 1: The Simulink window for 'RecurDyn_Host_Block_8_5' is shown. The 'Tools' menu is open, and 'Library Browser' is selected.

Step 2: The 'Simulink Library Browser' window is open, and 'gain' is entered in the search field. The search results show 'Gain' as the first result.

Step 3: The 'Gain' block is dragged from the library and placed into the Simulink diagram, connected to the 'RecurDyn Host Block'.

Step 4: The 'Function Block Parameters: Gain' dialog box is open, and the 'Gain' value is set to -1800.

Simulink Model - save

1. Select **File - Save As...**
2. Save as **invertedPendulum.mdl** as Simulink Models
3. **invertedPendulum.m** and **invertedPendulum.mdl** will be used for co-simulation

The screenshot shows the Simulink interface with the 'File' menu open and 'Save As...' selected. The 'Save As' dialog box is open, showing the file name 'invertedPendulum_MDL.mdl' and the file type 'Simulink Models (*.mdl)'. The file explorer below shows a list of files including 'invertedPendulum.m' and 'invertedPendulum_MDL.mdl'.

Name	Date modified	Type	Size
invertedPendulum.m	2018-06-19 2:00 PM	MATLAB Code	2 KB
invertedPendulum_MDL.mdl	2018-06-21 11:58	Simulink Model	34 KB

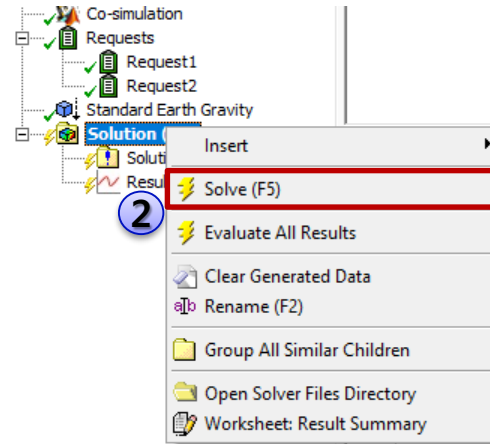


Simulation

1. Simulation
2. Post (Animation, Plot)

Simulation

1. **Save** the model before simulation
2. Select **Solve** to simulate the model
3. **Matlab** is automatically executed and **co-simulation is performed**.
4. After co-simulation is completed, green check icon is displayed. You can see the simulation log by clicking Solution Information.



A screenshot of the software interface showing three panels. The 'Outline' panel on the left shows a tree view with 'Solution (A6)' selected and 'Solution Information' highlighted with a red box and a blue circle with the number '4'. The 'Details of "Solution Information"' panel in the middle shows a table with the following data:

Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Identify Element Violations	0
Update Interval	2.5 s
Display Points	All

The 'Worksheet' panel on the right shows the 'Solver Output' section with the following text:

```
RecurDyn V9R1 [Windows x64 System 9.1.6951.0] Analysis message
Copyright (C) 1997 - 2017 FunctionBay, Inc. All rights reserved

Model file: MBDResult.rdyn [Model Name: MBDResult]

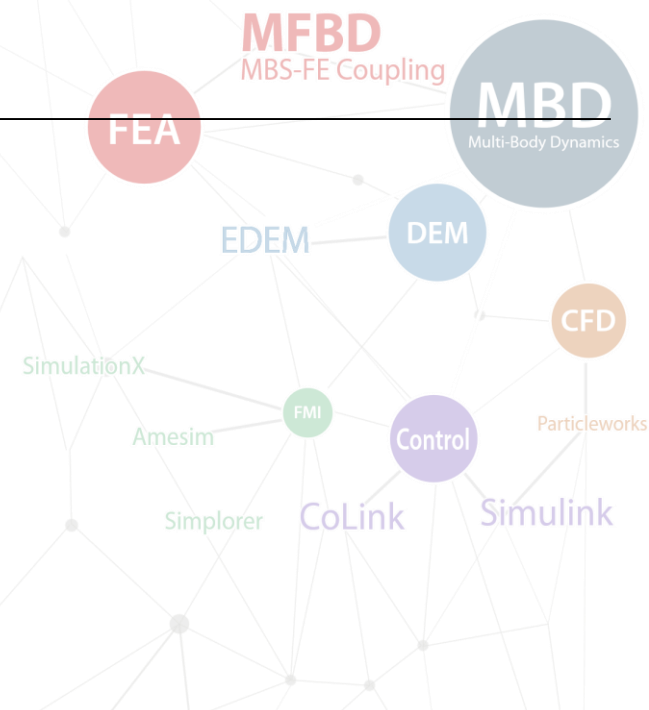
Model Verification Time Information
Build Date : Nov 21 2017
Build Time : 19:08:20

System Configuration Information
No. of Generalized Coordinate           = 2
No. of Generalized Velocity            = 2
No. of Rigid Body                       = 2
No. of Plant Input                      = 1
No. of Plant Output                     = 1

Kinematic Degree of Freedom             = 2

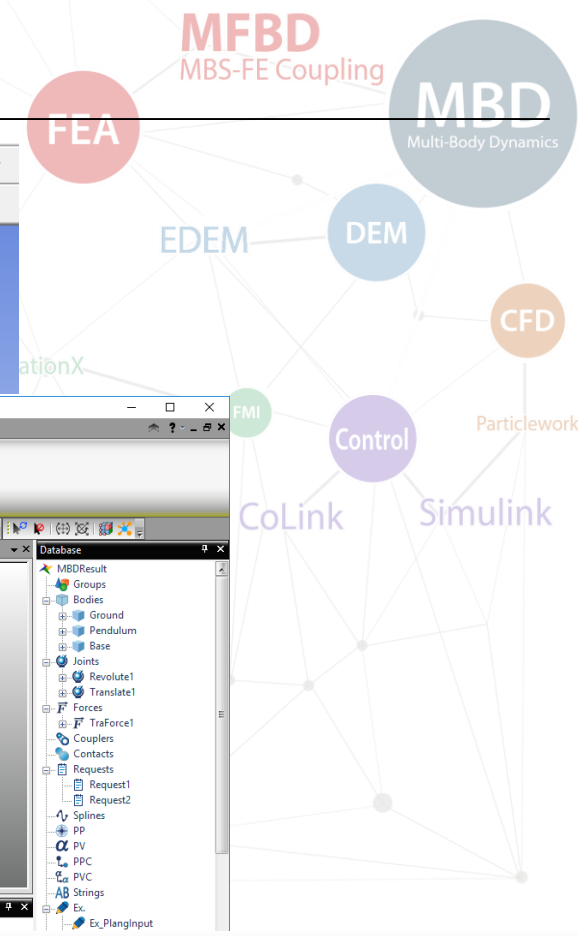
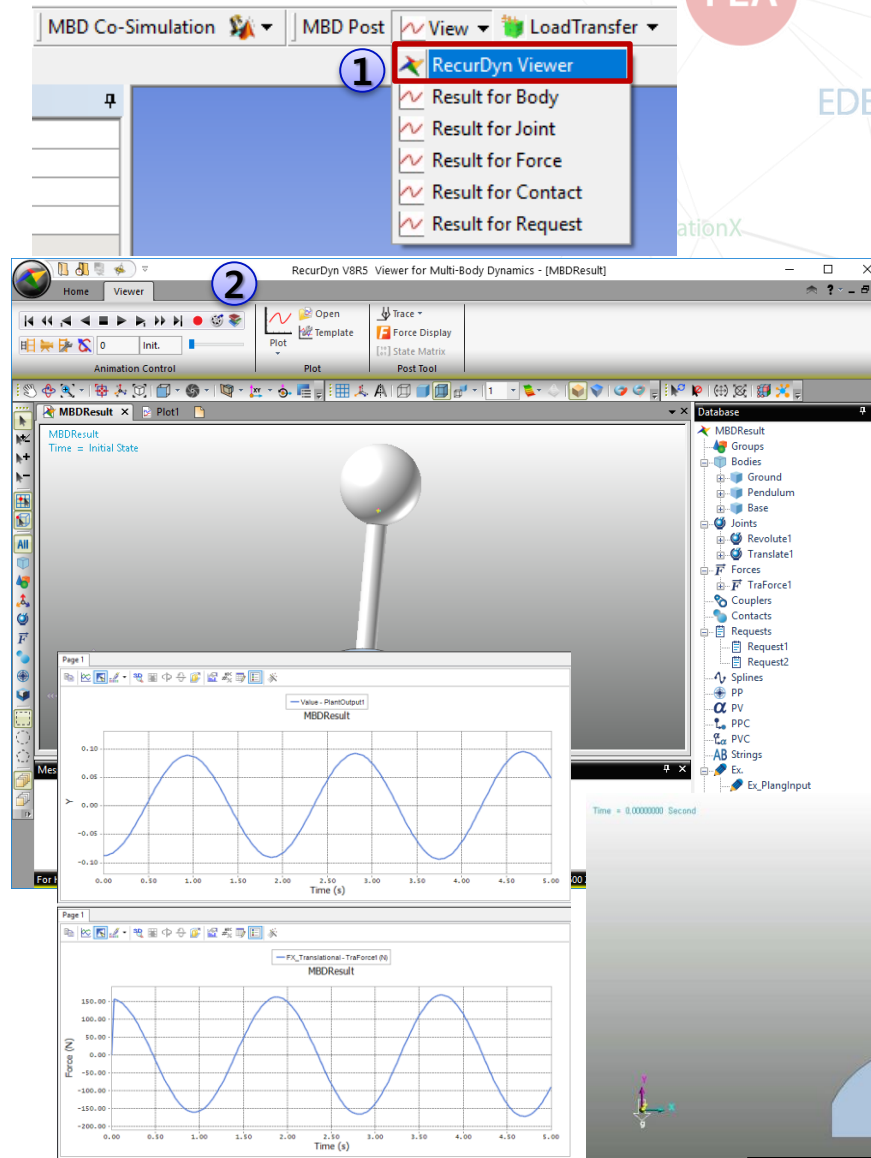
Total array size                         = 3972
Total memory size for array              = 0 MB

Success Process: Array Structure Construction
```

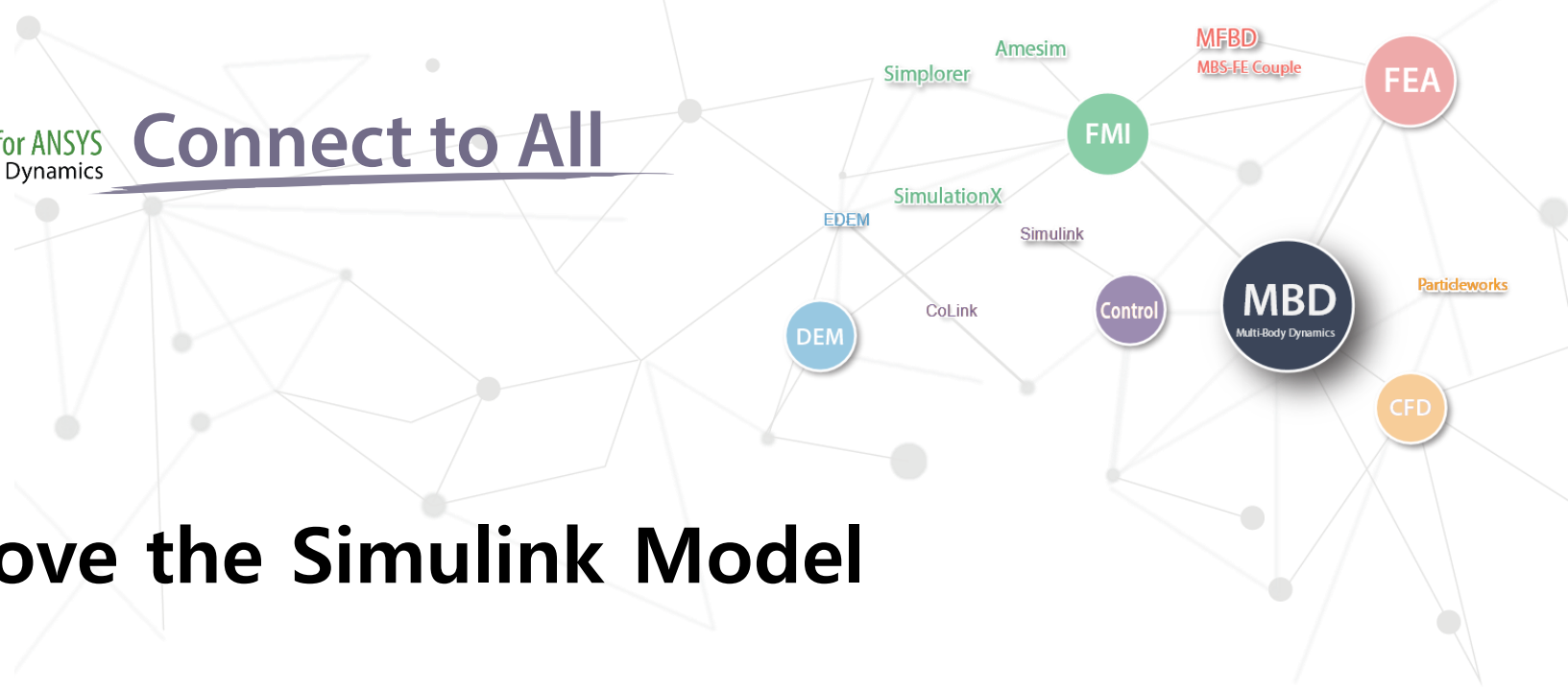


Post

1. In the MBD Post Toolbar, select View – **RecurDyn Viewer**
2. Review the results in RecurDyn Viewer
 - 1) Animation
 - 2) Plot



Play the animation



Improve the Simulink Model

1. Adding PID control block
2. POST

Adding PID controller

1. Execute Matlab
2. Set the current folder of Matlab to the full path of MBDResult
3. Input **invertedPendulum** in the **command window** + [Enter]
4. Input **makerd** in the **command window** + [Enter]
5. **Double click** **invertedPendulum_MDL.mdl** to open it.
6. Add **Integrator block** and **Derivative Block** and connect them and set Gains as below.
P Gain = -1800
I Gain = -250
D Gain = -200
7. **Save** the Simulink model
8. **Solve** the model from MBD for ANSYS

The screenshot shows the MATLAB R2015a environment. The Command Window displays the following commands and output:

```
>> invertedPendulum
----- RecurDyn & Matlab/SIMULINK -----
%%% INFO : RecurDyn plant actuators names :
1 PlantInput1
%%% INFO : RecurDyn plant sensors names :
1 PlantOutput1
=====

>> makerd
fx >> |
```

The File Explorer shows the current folder path: `D:\0000_WorkSpace\Training_Material\KIRO_Simulink_Training\SampleModels\InvertedPendulum\InvertPendulum_files\dp\`. The file `invertedPendulum_MDL.mdl` is highlighted in the file list.

A red box with the text "Double click" points to the `invertedPendulum_MDL.mdl` file in the File Explorer.

The Simulink model editor shows the `invertedPendulum_MDL` model. The model includes three gain blocks (Gain, Gain1, Gain2) connected to an Integrator block and a Derivative block, which are then connected to the RecurDyn Host Block. The gains are set as follows:

- Gain: -1800
- Gain1: -250
- Gain2: -200

Post

MFBD
MBS-FE Coupling

FEA

MBD
Multi-Body Dynamics

DEM

CFD

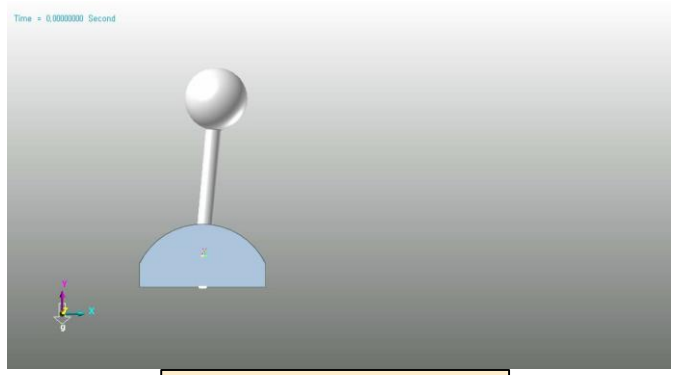
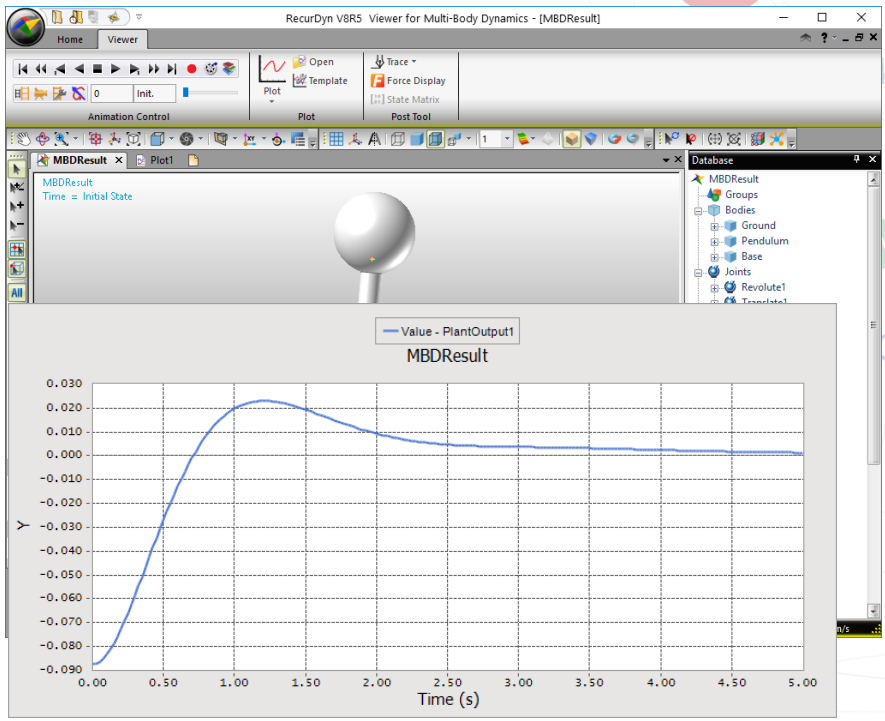
Particleworks

Control

CoLink

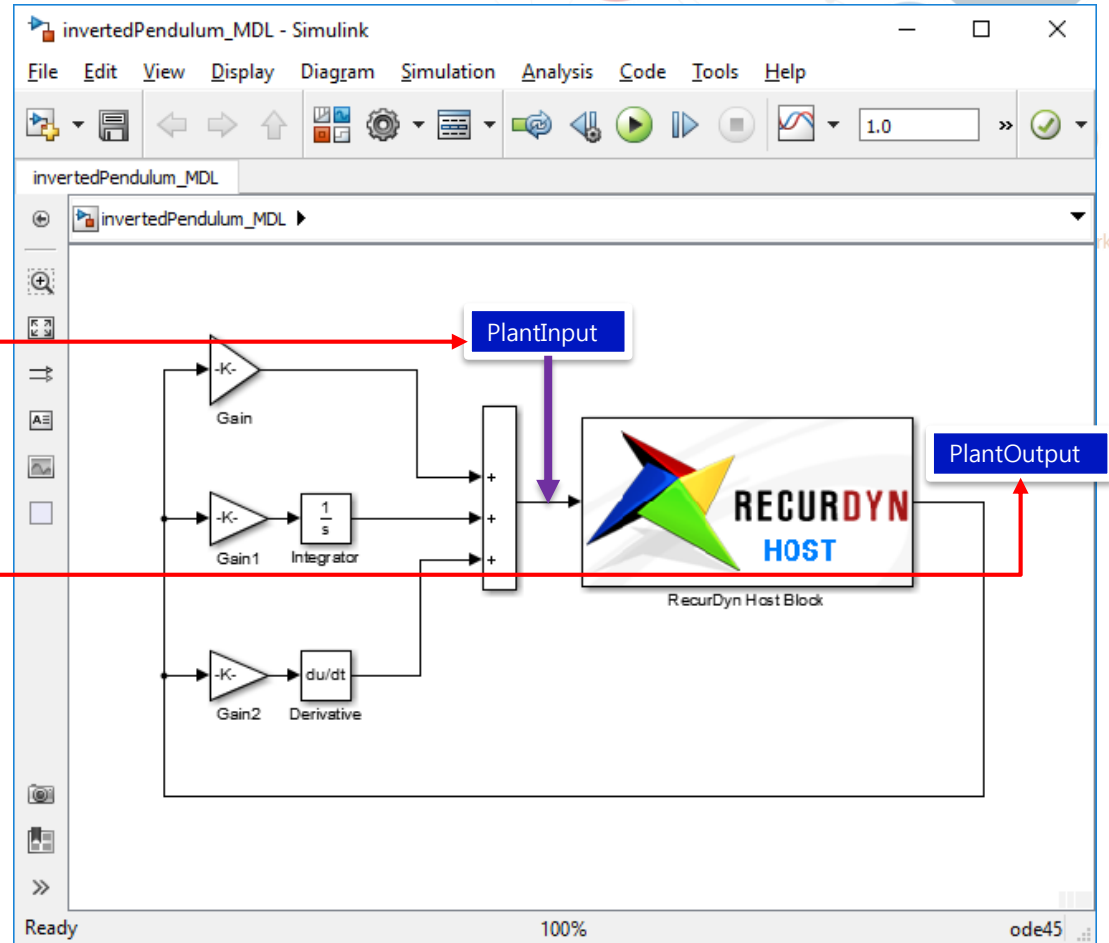
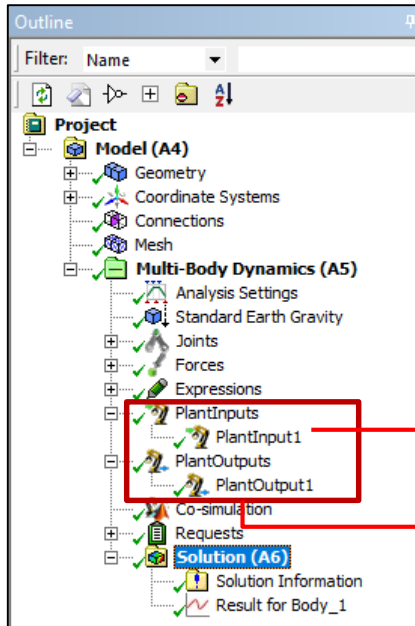
Simulink

1. In the MBD Post Toolbar, select View – **RecurDyn Viewer**
2. Review the results in RecurDyn Viewer
 - 1) Animation
 - 2) Plot



Play the animation

Explanation about the PID control model



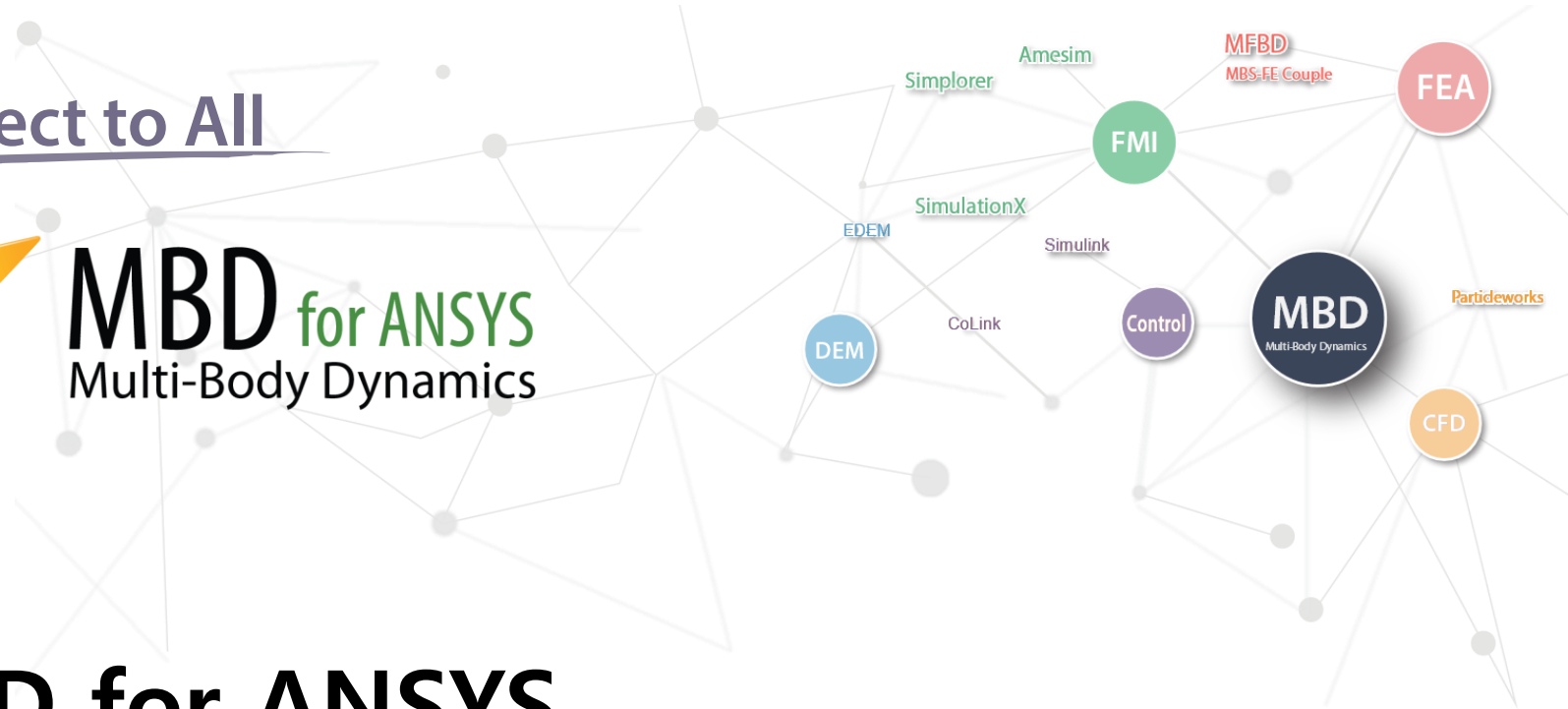
- The angle of the pendulum calculated by MBD for ANSYS is used as PlantOutput. It is used as an output of Host block.
- The output angle (PlantOutput) is used for the PID controller and the controlled value is used as PlantInput. It is used as an input of Host Block.

MBD for ANSYS Host Block = MBD for ANSYS Model

Connect to All



MBD for ANSYS
Multi-Body Dynamics



MBD for ANSYS Simulink Interface Training

강재묵

Korean

English

FunctionBay, Inc. Extended Application Team
Senior Manager (mbd4a@functionbay.co.kr)

□ MBD for ANSYS 모델링

1. MBD for ANSYS analysis project schematic 생성
2. CAD 파일 Import
3. Unit 설정 및 Gravity 생성
4. Joint 엔티티 생성
5. Force 엔티티 생성
6. Plant Input / output 생성
7. Analysis setting
8. Simulink Interface 생성

□ Matlab 모델링

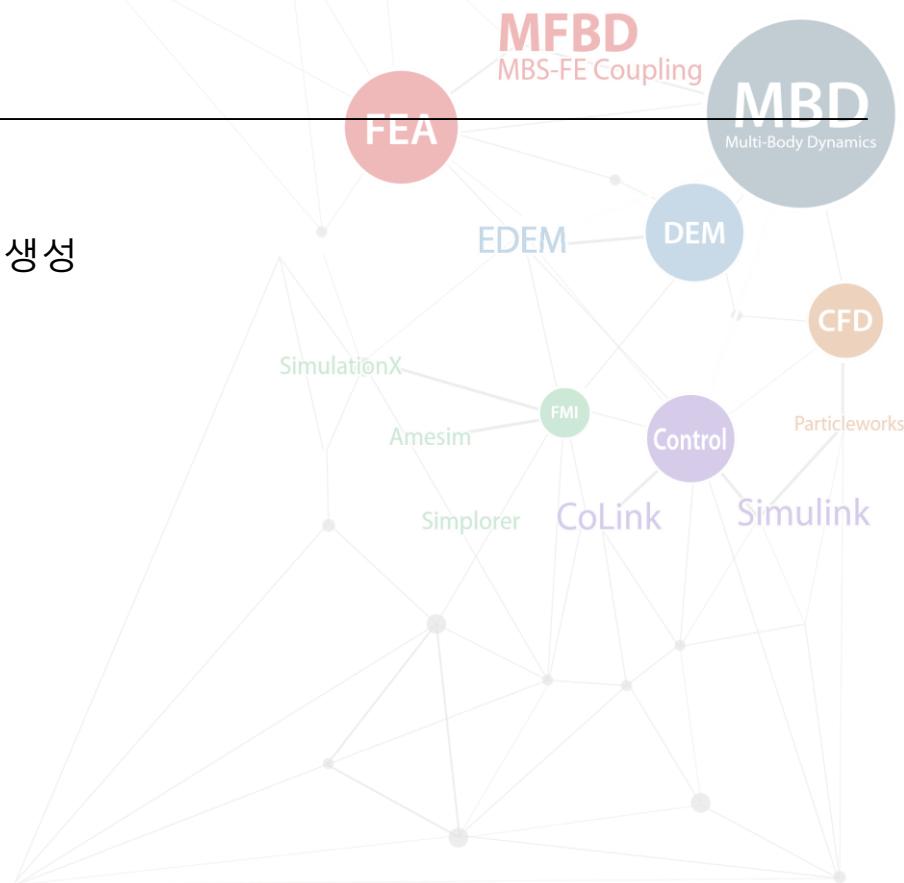
1. Current Directory 설정
2. RecurDyn Host Block 생성
3. Simulink 모델링

□ Simulation

1. Simulation
2. Post (Animation, Plot)

□ Simulink Model Upgrade

1. Adding PID control block
2. Post (Animation, Plot)





MBD for ANSYS 모델링

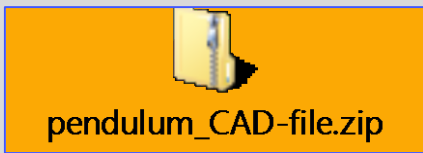
1. MBD for ANSYS analysis project schematic 생성
2. CAD 파일 Import
3. Unit 설정 및 Gravity 생성
4. Joint 엔티티 생성
5. Force 엔티티 생성
6. Plant Input / output 생성
7. Analysis setting
8. Simulink Interface 생성

Creation MBD for ANSYS Analysis

MFBFD
MBS-FE Coupling

MBD
Multi-Body Dynamics

1. Toolbox 에서 Multi-Body Dynamics analysis 를 생성
2. Project Schematic 에서 **Geometry** 셀을 마우스 우 클릭
3. Pop up menu 에서 **Import Geometry – Browse** 를 클릭
4. **pendulum.x_t** 를 선택하여 Import
5. Geometry 셀에서 마우스 우 클릭 메뉴로 **Edit Geometry in DesignModeler** 를 클릭



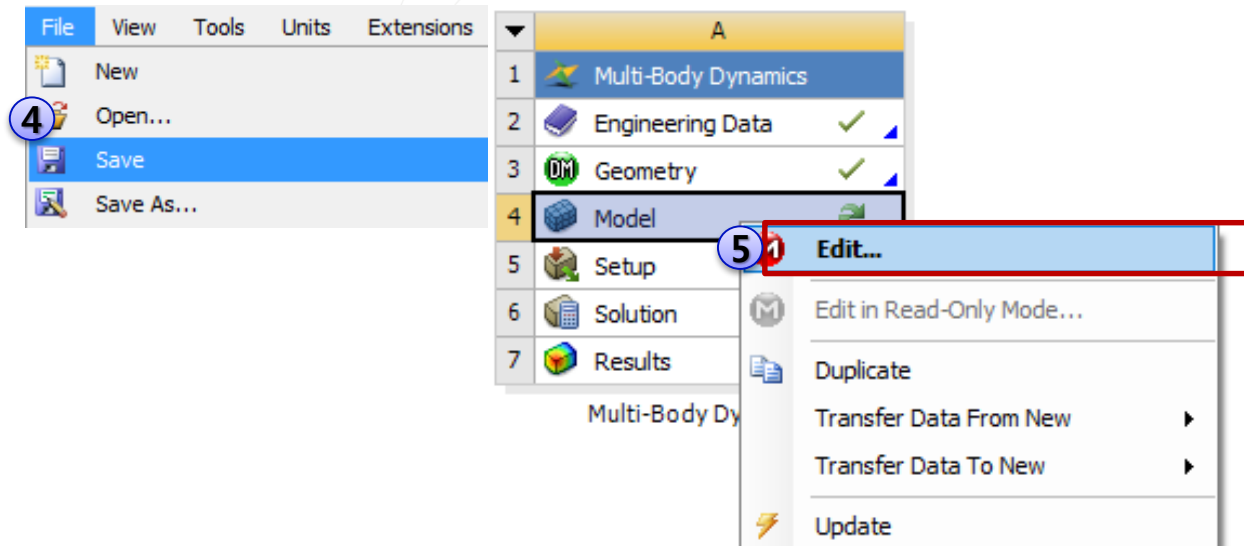
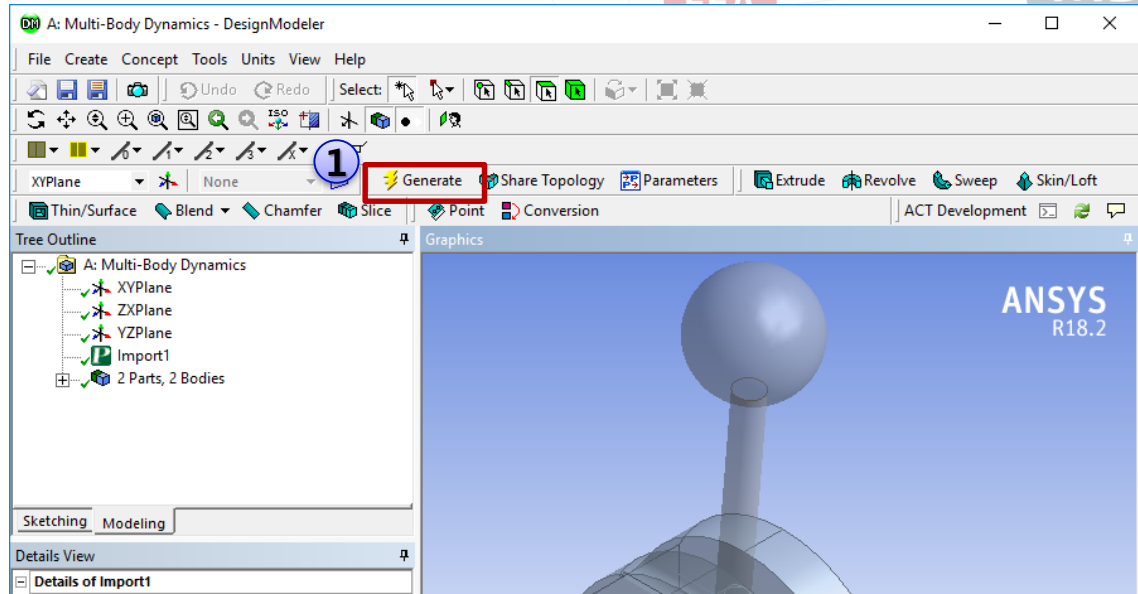
The image shows a series of screenshots from the ANSYS Workbench interface, illustrating the steps to create a Multi-Body Dynamics (MBD) analysis. The steps are numbered 1 through 5:

- Step 1:** The **Toolbox** on the left is shown with **Multi-Body Dynamics** selected and highlighted with a red box and a circled '1'.
- Step 2:** The **Project Schematic** on the right shows the **Geometry** cell selected and highlighted with a red box and a circled '2'. A red arrow points to a text box that says "마우스 우 클릭" (Right-click mouse).
- Step 3:** A context menu is shown over the **Geometry** cell. The **Import Geometry** option is expanded, and **Browse...** is selected, highlighted with a red box and a circled '3'.
- Step 4:** A file selection dialog is shown with **pendulum.x_t** selected and highlighted with a red box and a circled '4'.
- Step 5:** The context menu is shown again, with **Edit Geometry in DesignModeler...** selected and highlighted with a red box and a circled '5'.

In the background, there is a network diagram showing connections between various simulation tools: FEA, MFBFD, MBS-FE Coupling, MBD, DEM, CFD, Particleworks, Simulink, and CoLink.

Import CAD file

1. DesignModeler 앱이 실행되면 **Generate** 메뉴를 클릭 합니다.
2. 생성된 Geometry 를 확인 합니다.
3. DesignModeler 앱을 종료 합니다.
4. 프로젝트를 저장(**Save**)합니다.
5. Project Schematic에서 **Model** 셀의 마우스 우클릭 메뉴에서 **Edit**를 누릅니다.



Set unit, change geometry names, creation gravity

1. Units 메뉴에서 **Metric (mm, kg, N, s, mV, mA)** 로 변경
2. Outline 에서 Geometry 이름 변경. **Pendulum, Base**
3. Outline의 Multi-Body Dynamics에서 마우스 우 클릭 후, 팝업 메뉴에서 **Insert – Standard Earth Gravity** 클릭
4. Direction 을 **-Y Direction** 으로 변경

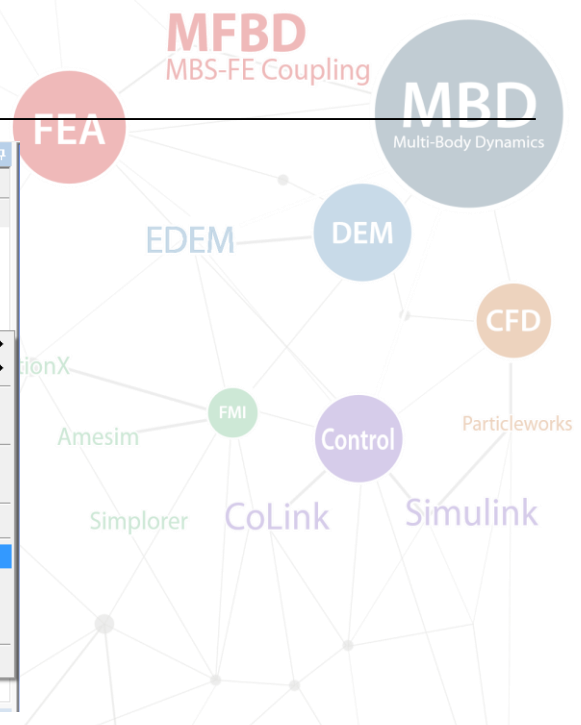
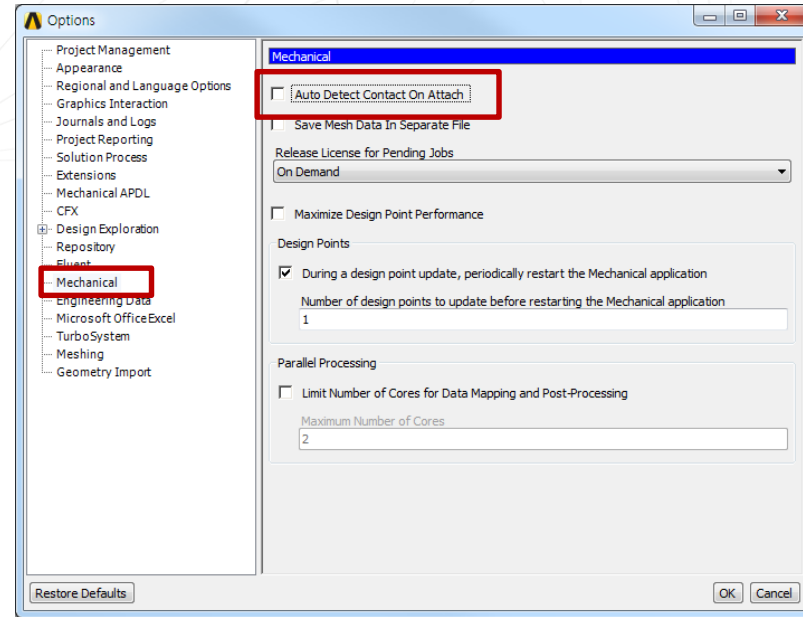
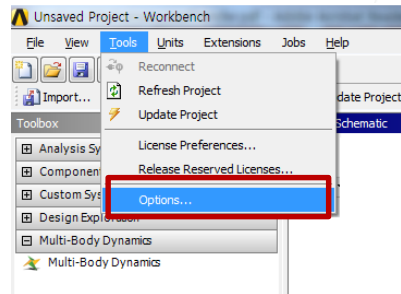
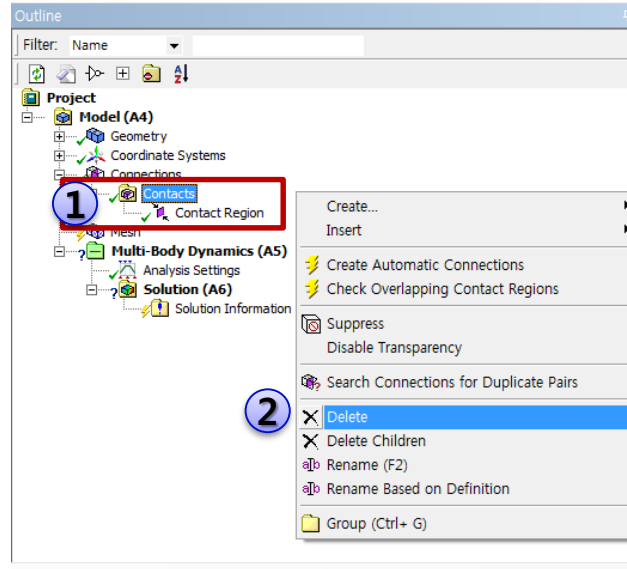
The screenshot shows the ANSYS Mechanical Enterprise interface with several key actions highlighted by numbered circles and red boxes:

- 1:** The **Units** menu is open, and **Metric (mm, kg, N, s, mV, mA)** is selected.
- 2:** In the **Outline** pane, the **Geometry** folder is expanded, and the names of **Pendulum** and **Base** are being edited.
- 3:** A right-click context menu is open over the **Multi-Body Dynamics (A5)** folder, and **Insert > Standard Earth Gravity** is selected.
- 4:** The **Details of "Standard Earth Gravity"** panel is shown, with the **Direction** dropdown menu set to **-Y Direction**.

Scope	
Geometry	All Bodies
Definition	
Coordinate System	Global Coordinate System
X Component	0. mm/s ² (ramped)
Y Component	-9806.6 mm/s ² (ramped)
Z Component	0. mm/s ² (ramped)
Suppressed	No
Direction	-Y Direction

Delete Contact Region

1. Outline 윈도우에서 Contacts 선택
2. 팝업메뉴에서 **Delete** 선택
3. 이 작업을 이후에는 반복하지 않기 위해서는 Mechanical의 **Auto Detect Contact on Attach** 옵션의 체크를 해제. (Workbench 에서 **Tools – Options...**)



Joint Creation – Revolute Joint

1. MBD Entities Toolbar에서 Joint – **Revolute** 클릭
2. Detail 창에서 Connection Type 을 **Body-Body** 로 변경
3. Base Body 로 '**Base**' Geometry 선택
4. Action Body 로 '**Pendulum**' Geometry 선택
5. Origin 은 **0,0,0** 으로 설정
6. Rotational Axis로 **0,0,1**으로 설정

MBD Entities **Joints** Forces Contacts Expressions

① Fixed

② **Revolute**

Translate

Planar

Cylindrical

Spherical

Screw

Universal

Con.Veloc

CMotion

PointCurve

CurveCurve

Coupler

Gear

Details of "Revolute1"

② **Definition**

Connection Type **Body-Body**

Coordinate System No

③ **Base Body Information**

Scoping Method Geometry Selection

Scope 1 Body

Body **Base**

④ **Action Body Information**

Scoping Method Geometry Selection

Scope 1 Body

Body **Pendulum**

⑤ **Origin**

Origin Click to Change

Origin X 0 [mm]

Origin Y 0 [mm]

Origin Z 0 [mm]

⑥ **Rotational Axis**

Axis Click to Change

Axis X 0

Axis Y 0

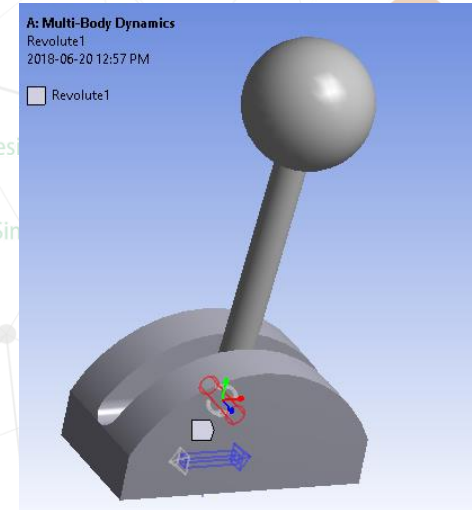
Axis Z 1

Motion

Include Motion No

Friction

Include Friction No



Joint Creation – Translational Joint

1. MBD Entities Toolbar에서 Joint – **Translate** 클릭
2. Detail 창에서 Connection Type 을 **Ground-Body** 로 변경
3. Action Body 로 '**Base**' Geometry 선택
4. Origin 은 **0,-100,0** 으로 설정
5. Translational Axis로 **1,0,0** 으로 설정

MBD Entities **Joints** Forces Contacts Expressions

Fixed

Revolute

Translate

Planar

Cylindrical

Spherical

Screw

Universal

Con.Veloc

CMotion

PointCurve

CurveCurve

Coupler

Gear

Details of "Translate1"

Definition

Connection Type: Ground-Body

Coordinate System: No

Base Body Information

Body: Ground

Action Body Information

Scoping Method: Geometry Selection

Scope: 1 Body

Body: Base

Origin

Origin: Click to Change

Origin X: 0 [mm]

Origin Y: -100 [mm]

Origin Z: 0 [mm]

Translational Axis

Axis: Click to Change

Axis X: 1

Axis Y: 0

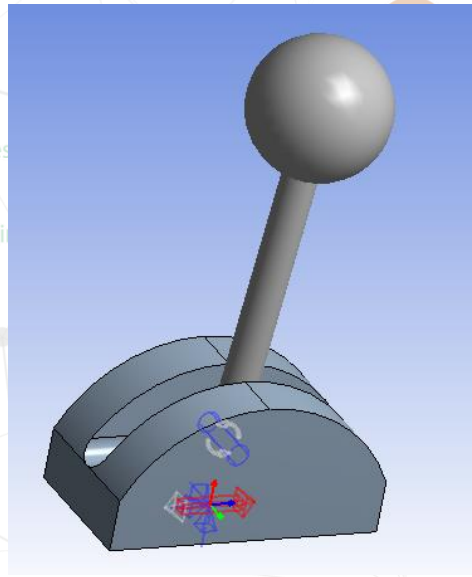
Axis Z: 0

Motion

Include Motion: No

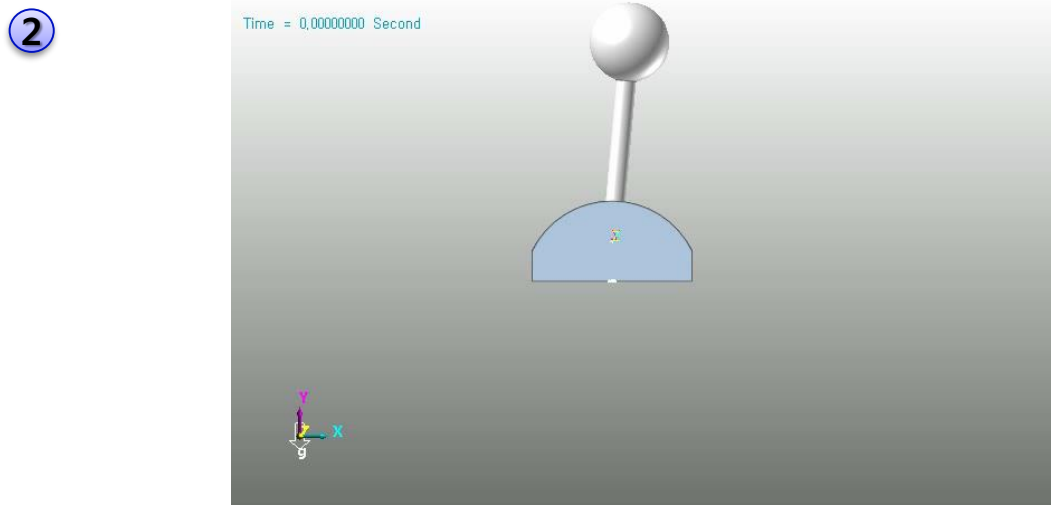
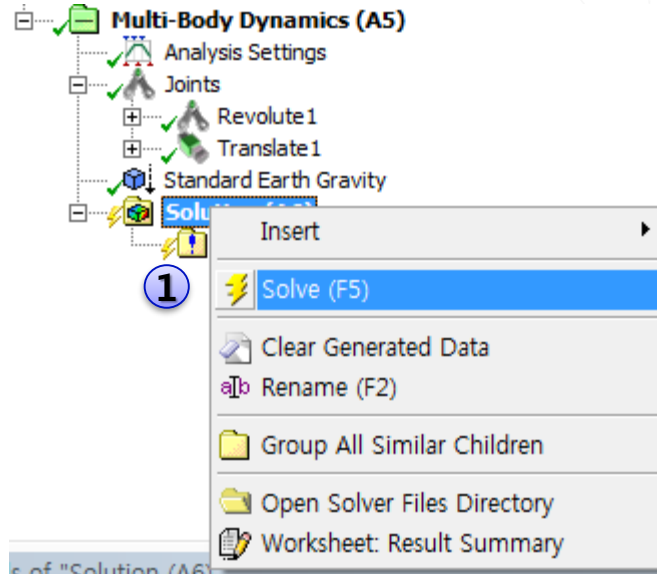
Friction

Include Friction: No

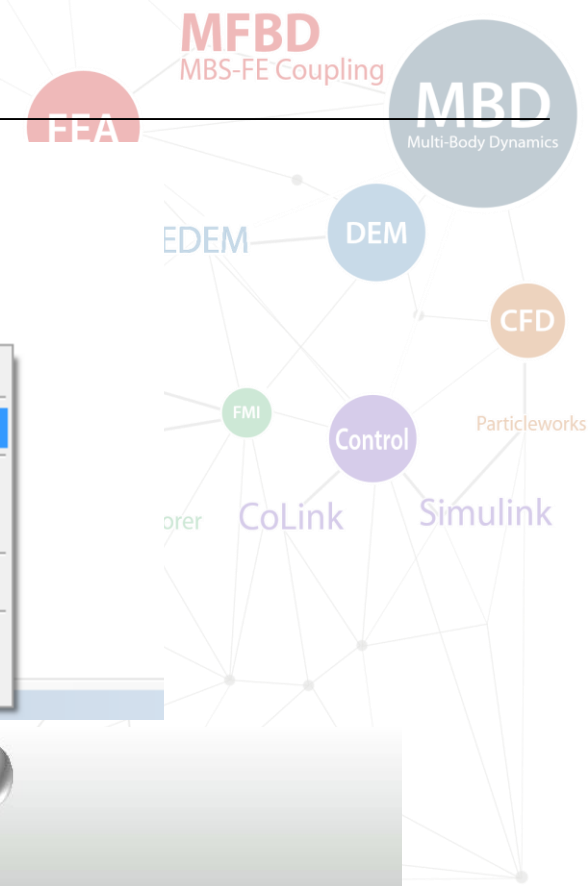


Simulation

1. **Solve** 를 통해 시뮬레이션
2. 애니메이션을 확인 가능
3. 다음 페이지에서는 Pendulum을 위한 PID제어기를 추가할 것입니다.

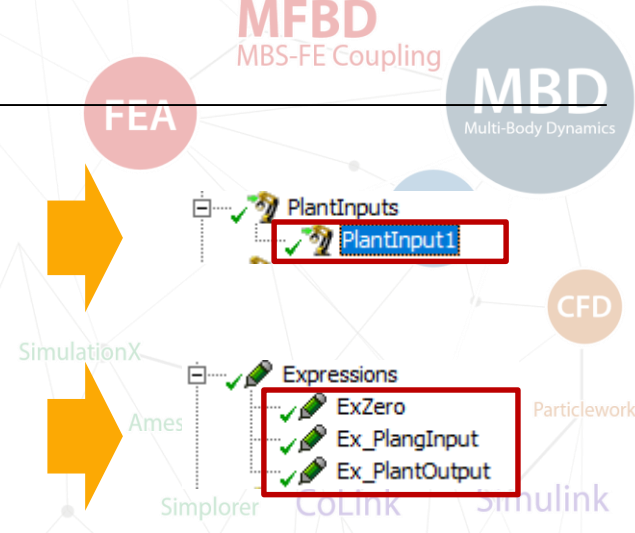
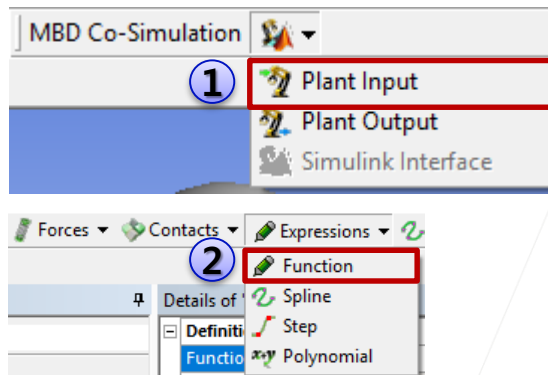


Play the animation

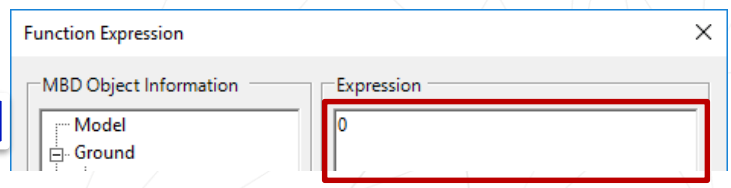


Plant Input, Expression 생성

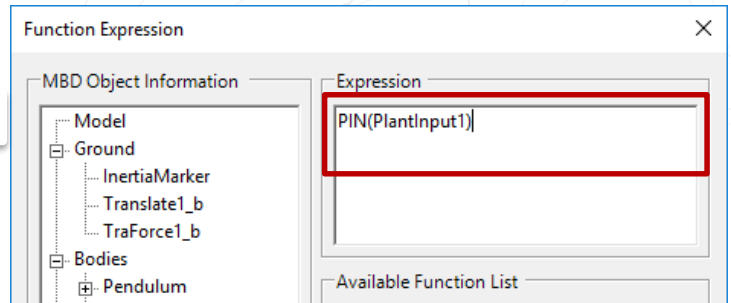
1. MBD Co-Simulation Toolbar에서 **Plant Input** 클릭
2. MBD Entities Toolbar에서 Expressions – **Function** 클릭
3. Expression 이름을 **ExZero** 로 변경하고, Function Expression을 2개 더 만든 후, **Ex_PlantInput**, **Ex_PlantOutput**으로 이름 변경
4. ExZero 입력창에 **0** 입력
5. Ex_PlantInput 입력창에 **PIN(PlantInput1)** 입력
6. Ex_PlantOutput 입력창에 **(AZ(Pendulum.Revolute1_a, Base.Revolute1_b)-5D)** 입력



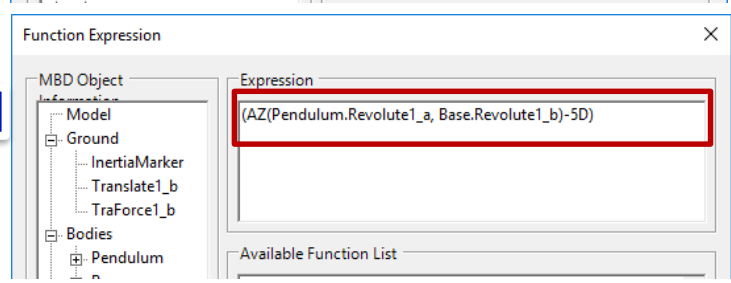
3
ExZero



4
Ex_PlantInput

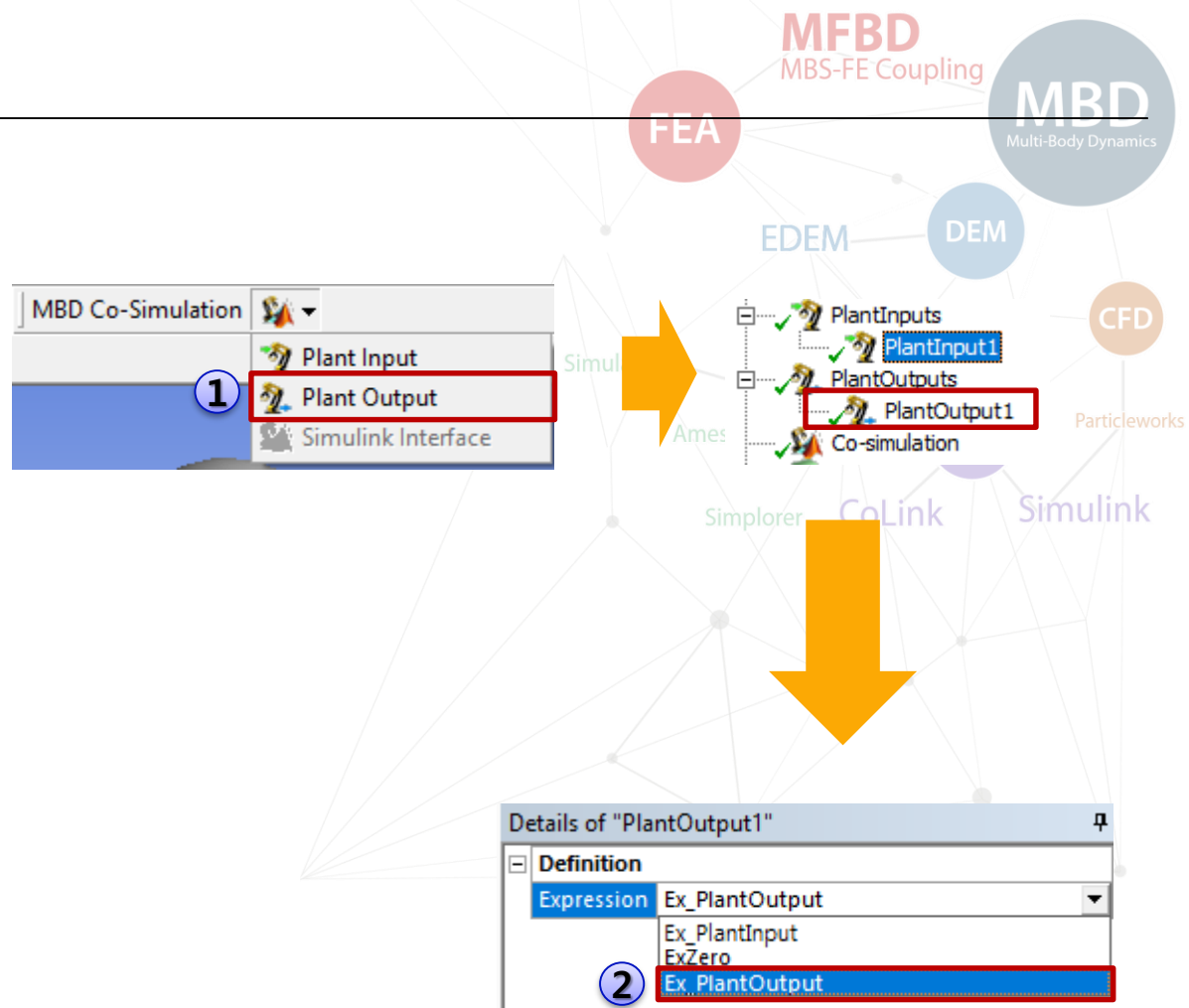


5
Ex_PlantOutput



PlantOutput 생성

1. MBD Co-Simulation Toolbar에서 **Plant Output** 클릭
2. Detail 창 에서 Expression 으로 **Ex_PlantOutput** 선택



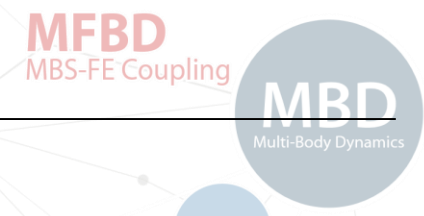
Force Creation – Translational Force

1. MBD Entities Toolbar 에서 Force – **Tra.Force** 클릭
2. Detail 창 에서 Connection Type 으로 **Ground-Body** 선택
3. Action Body 로 '**Base**' Geometry 선택
4. Origin 을 **0,-100,0** 으로 설정
5. Force Expression 에서 FX Function Exp 로 **Ex_PlantInput** 선택, 나머지는 **ExZero** 선택

The screenshot shows the software interface for creating a translational force. The 'Forces' toolbar is highlighted with a red box. The tree view shows 'Multi-Body Dynamics (A5)' with 'TraForce1' selected under the 'Forces' folder. The 'Details of TraForce1' panel is open, showing the following configuration:

Definition	
Connection Type	Ground-Body
Coordinate System	No
Base Body Information	
Body	Ground
Action Body Information	
Scoping Method	Geometry Selection
Scope	1 Body
Body	Base
Origin	
Origin	Click to Change
<input type="checkbox"/> Origin X	0 [mm]
<input type="checkbox"/> Origin Y	-100 [mm]
<input type="checkbox"/> Origin Z	0 [mm]
Force	
FX Function Exp	Ex_PlantInput
FY Function Exp	ExZero
FZ Function Exp	ExZero
Reference Marker	
Use Reference Marker	No

Analysis Settings



1. Analysis Settings 의 Details 에서 다음과 같이 설정
EndTime = 5
Step = 200
Maximum Time Step = 0.001

Outline

Filter: Name

Project

- Model (A4)
 - Geometry
 - Pendulum
 - Base
 - Coordinate Systems
 - Connections
 - Mesh
 - Multi-Body Dynamics (A5)**
 - Analysis Settings**
 - Joints
 - Revolute 1
 - Translate 1
 - Forces
 - TraForce 1
 - Expressions
 - ExZero
 - Ex_PlantInput

Details of "Analysis Settings"

Definition

Simulation Type	Dynamics
-----------------	----------

General

<input type="checkbox"/> End Time	5 [sec]
<input type="checkbox"/> Step	200

Parameter

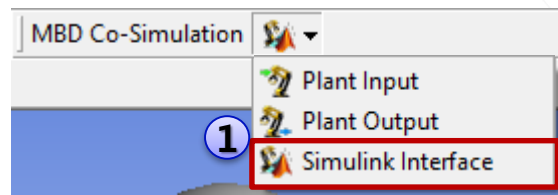
Maximum Order	2
<input type="checkbox"/> Maximum Time Step	0.0001
<input type="checkbox"/> Initial Time Step	1E-06
<input type="checkbox"/> Error Tolerance	0.005
Integrator Type	IMGALPHA
Numerical Damping	1
Constant Stepsize	1E-05
Jacobian Evaluation	100
Stop Condition	No

Include

Static Analysis	No
-----------------	----

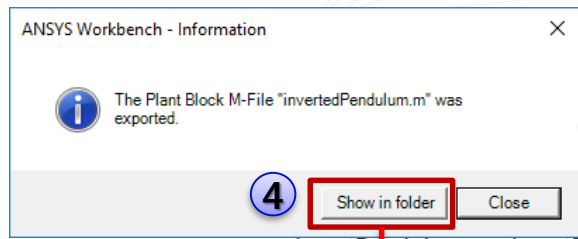
Simulink Interface

1. MBD Co-Simulation
Toolbar에서 **Simulink Interface** 클릭
2. 아래와 같이 설정
Interface Version = **2.0**
Sampling Period = **0.001**
(MATLAB 2016a 이상의 버전을 쓴다면 Interface Version = 3.0)
3. Plant Block M-File filename 을 invertedPendulum으로 설정 후, **"Click to Export M-File"** 클릭
4. Show in folder를 눌러 **invertedPendulum.m** 파일이 생성 되었음을 확인.
(MBD result 폴더에 있어야 함)
5. **Click to Change**를 눌러 Matlab Executable Location 으로 Matlab.exe 위치 설정
(\$Matlabroot\bin\win**\MATLAB.exe)
6. Simulink Model filename 으로 **invertedPendulum_MDL** 설정
(MBD result 폴더에 있어야 함)



Details of "Co-simulation"

Definition	
Interface Version	2.0
<input type="checkbox"/> Sampling Period (Control Time Step)	0.001
<input type="checkbox"/> Waiting Time (seconds)	60
Export Plant Block M-File	Click to Export M-File
Plant Block M-File filename	invertedPendulum
Set Matlab Executable Location	Click to Change
Matlab Executable Location	C:\Program Files\MATLAB\R2015a\bin\win64\MATLAB.exe
Set Simulink Model filename	Click to Change
Simulink Model filename	invertedPendulum_MDL



> InvertPendulum > InvertPendulum_files > dp0 > SYS > MECH > MBDResult

Name	Date modified	Type
invertedPendulum.m	2018-06-19 2:00 PM	MATLAB Code
invertedPendulum_m.m	2018-06-20 12:43	MATLAB Code
invertedPendulum_MDL.mdl	2017-04-24 3:05 PM	Simulink Model



Matlab 모델링

1. Current Directory 설정
2. RecurDyn Host Block 생성
3. Simulink 모델링

Set working directory

1. Outline 에서 Solutions 의 마우스 우클릭 메뉴에서 **Open Solver Files Directory** 클릭
2. 실행된 탐색기에서 **MBDResult** 폴더 클릭
3. **invertedPendulum.m** 파일이 존재하는 **폴더 경로** 를 복사
4. 3번에서 복사한 경로를 Matlab의 **Current Folder**에 붙여 넣고 엔터

The screenshot illustrates the steps to set the working directory in MATLAB. It shows the Simulink model tree on the left, a file explorer window in the middle, and the MATLAB R2015a interface at the bottom.

Step 1: In the Simulink model tree, the 'Solution (A6)' folder is right-clicked, and the context menu is shown. The 'Open Solver Files Directory' option is highlighted with a red box and a blue circle labeled '1'.

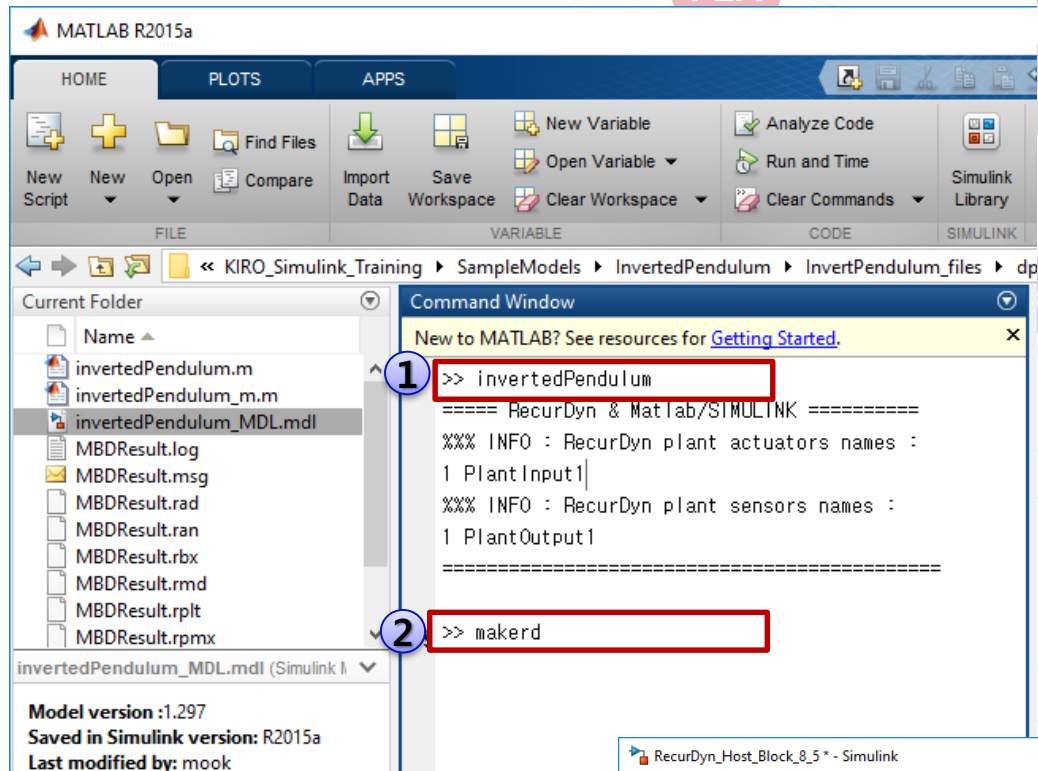
Step 2: The file explorer shows the path: SampleModels > InvertedPendulum > InvertPendulum_files > dp0. The 'MBDResult' folder is highlighted with a red box and a blue circle labeled '2'.

Step 3: The file explorer shows the path: Training_Material\KIRO_Simulink_Training\SampleModels\InvertedPendulum\InvertPendulum_files\dp0\SYS\MECH\MBDResult. The path is highlighted with a red box and a blue circle labeled '3'.

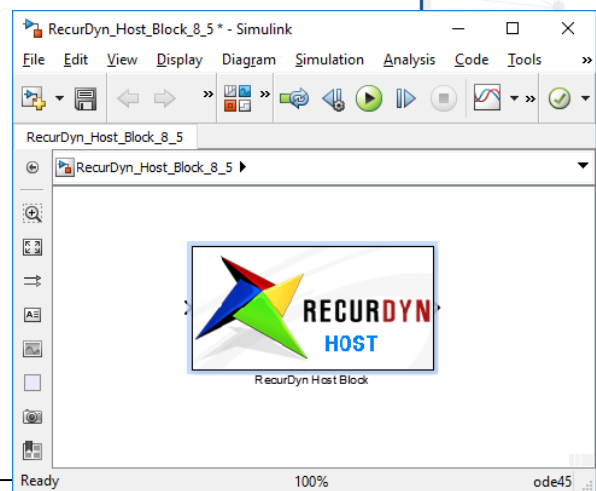
Step 4: The MATLAB R2015a interface shows the 'Current Folder' field in the Command window. The path from step 3 is pasted into the field, highlighted with a red box and a blue circle labeled '4'.

MBD for ANSYS Host Block 생성

1. Command Window 에 **invertedPendulum** 을 입력하고 엔터
2. Command Window 에 **makerd** 를 입력하고 엔터
3. Simulink Block window 가 실행되면서 RecurDyn Host Block 이 생성



3
Makerd 를 입력하면 오른쪽 그림과 같이 RecurDyn Host Block 이 생성된다.



Simulink Block 생성

1. Simulink Window 에서 **Tools – Library Browser** 클릭
2. Simulink Library Browser 에서 **gain** 입력 하여 검색
3. Gain Block 을 RecurDyn Host Block 에 있는 위치로 **Drag & Drop**
4. Gain 의 값을 **-1800** 으로 설정

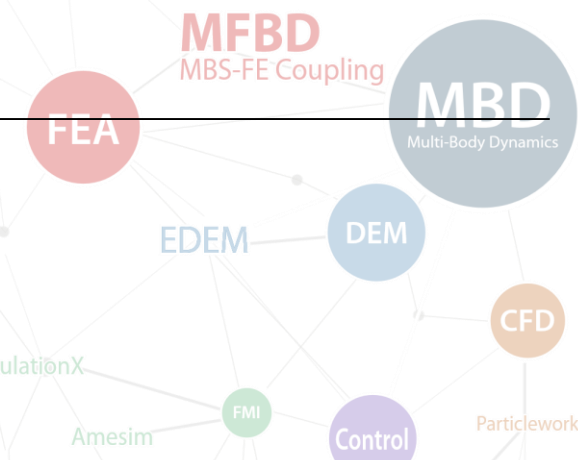
1

2

3

4

Simulink 모델 저장



1. Save As... 를 클릭
2. 파일명 **invertedPendulum.mdl**
Save as type : Simulink Models (*.mdl) 으로 저장
3. invertedPendulum.m 과
invertedPendulum.mdl 파일이
최종적으로 Co-Simulation 에
사용됨

1

2

3

Name	Date modified	Type	Size
invertedPendulum.m	2018-06-19 2:00 PM	MATLAB Code	2 KB
invertedPendulum_MDL.mdl	2018-06-21 11:58	Simulink Model	34 KB

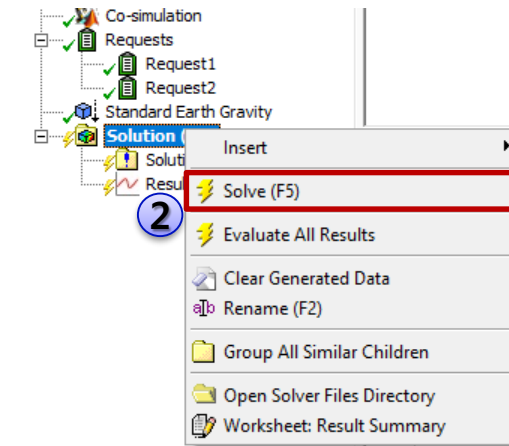


Simulation

1. Simulation
2. Post (Animation, Plot)

Simulation

1. Simulation 하기 전에 모델 저장
2. Outline 에서 Solution 마우스 우 클릭 메뉴에서 **Solve** 클릭
3. Matlab 이 자동으로 실행되면서 Co-simulation
4. 해석이 완료 되면 녹색 체크 아이콘으로 바뀌고, Solution Information 을 클릭 하면 해석 결과에 대한 로그를 볼 수 있음



Outline

Filter: Name

- Project
- Model (A4)
 - Geometry
 - Coordinate Systems
 - Connections
 - Mesh
 - Multi-Body Dynamics (A5)
 - Analysis Settings
 - Standard Earth Gravity
 - Joints
 - Forces
 - Expressions
 - PlantInputs
 - PlantOutputs
 - Co-simulation
 - Requests
 - Solution (A6)
 - Solution Information**
 - Result for Body_1

Details of "Solution Information"

Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Identify Element Violations	0
Update Interval	2.5 s
Display Points	All

Worksheet

Solver Output

RecurDyn V9R1 [Windows x64 System 9.1.6951.0] Analysis message
Copyright (C) 1997 - 2017 FunctionBay, Inc. All rights reserved

Model file: MBDResult.rdyn [Model Name: MBDResult]

Model Verification Time Information

Build Date : Nov 21 2017
Build Time : 19:08:20

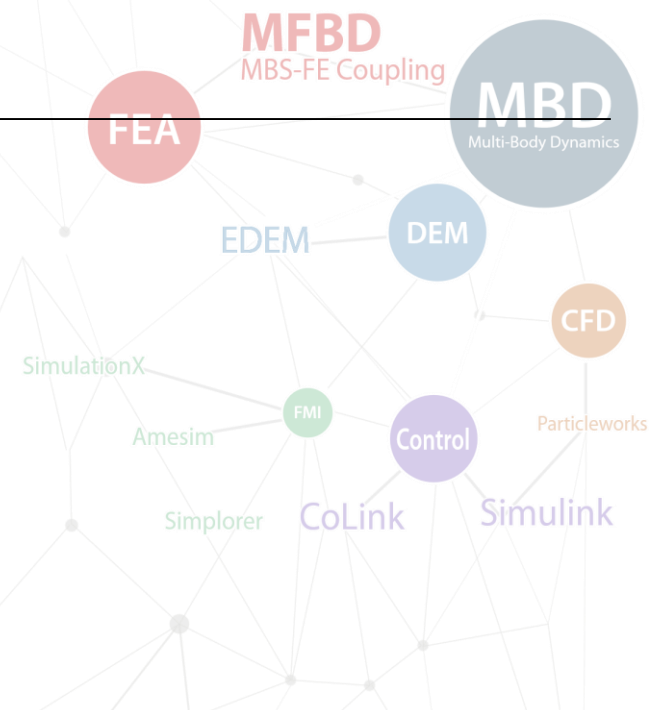
System Configuration Information

No. of Generalized Coordinate	= 2
No. of Generalized Velocity	= 2
No. of Rigid Body	= 2
No. of Plant Input	= 1
No. of Plant Output	= 1

Kinematic Degree of Freedom = 2

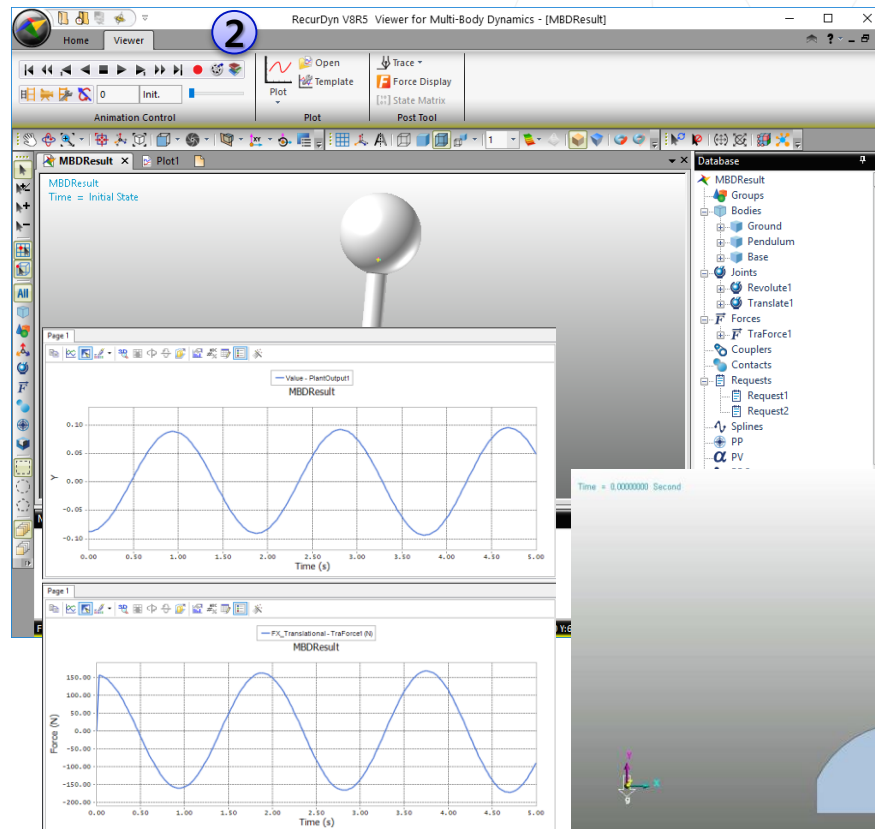
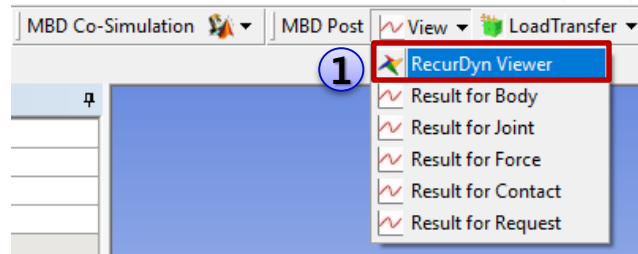
Total array size = 3972
Total memory size for array = 0 MB

Success Process: Array Structure Construction

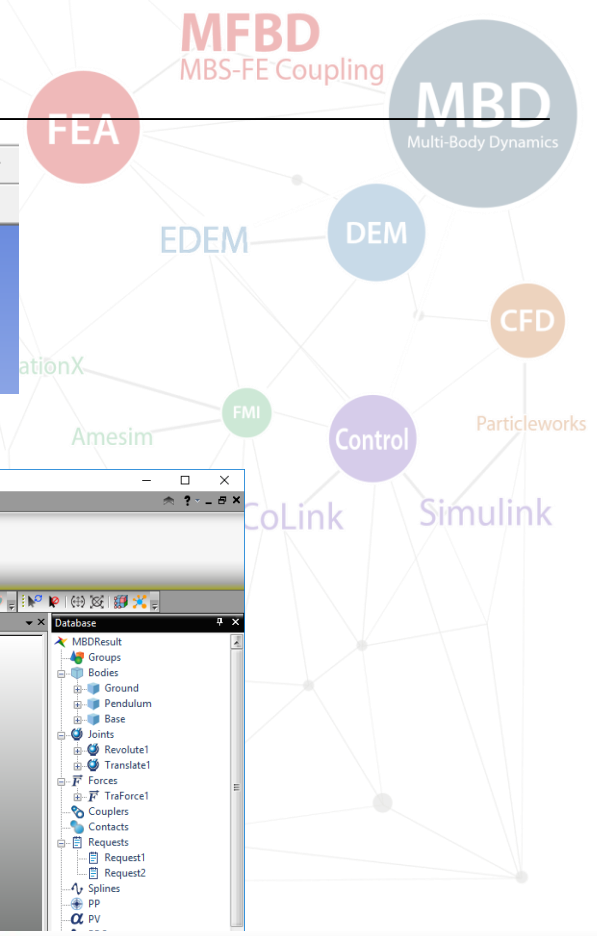


Post

1. MBD Post Toolbar 에서 View – **RecurDyn Viewer** 클릭
2. RecurDyn Viewer 에서 POST 확인
 - 1) 애니메이션
 - 2) Plot



Play the animation





Simulink Model Upgrade

1. Adding PID control block
2. POST

Adding PID controller

1. Matlab을 실행
2. MBDResult 폴더 경로를 Matlab의 Current Folder로 설정
3. Command Window 에 **invertedPendulum** 입력하고 엔터
4. **Makerd** 입력하고 엔터
5. **invertedPendulum_M0|.m0|** 을 더블 클릭하여 오픈
6. Library Browser에서 **Integrator block, Derivative Block** 추가한 후, 그림과 같이 Block 연결하고 Gain값을 설정
 P Gain = -1800
 I Gain = -250
 D Gain = -200
7. Simulink 모델 **Save**
8. MBD for ANSYS에서 모델을 다시 시뮬레이션 (**Solve**)

The screenshot shows the MATLAB R2015a environment. The Command Window contains the following commands:

```
>> invertedPendulum
===== RecurDyn & Matlab/SIMULINK =====
%%% INFO : RecurDyn plant actuators names :
1 PlantInput1
%%% INFO : RecurDyn plant sensors names :
1 PlantOutput1
=====

>> makerd
fx >> |
```

The File Explorer shows the current folder path: `D:\0000_WorkSpace\Training_Material\KIRO_Simulink_Training\SampleModels\InvertedPendulum\InvertPendulum_files\dp\`. The file `invertedPendulum_MDL.mdl` is highlighted in the file list.

A red box with the text "Double click" points to the `invertedPendulum_MDL.mdl` file in the File Explorer.

The Simulink model editor shows the `invertedPendulum_MDL` model. The model includes a `RecurDyn Host Block` and three gain blocks (Gain, Gain1, Gain2) connected to an Integrator and Derivative block. The gain values are set as follows:

- Gain: -1800
- Gain1: -250
- Gain2: -200

Post

MFBD
MBS-FE Coupling

FEA

MBD
Multi-Body Dynamics

DEM

CFD

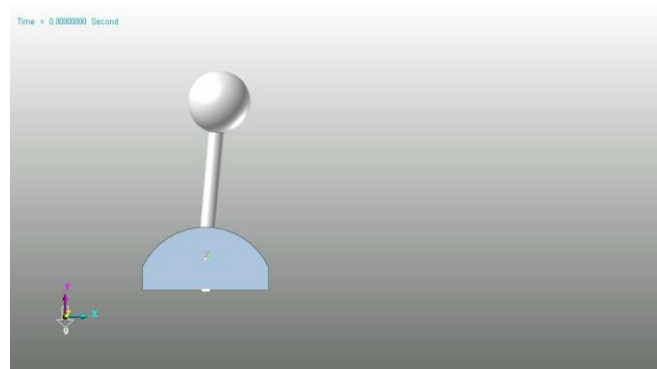
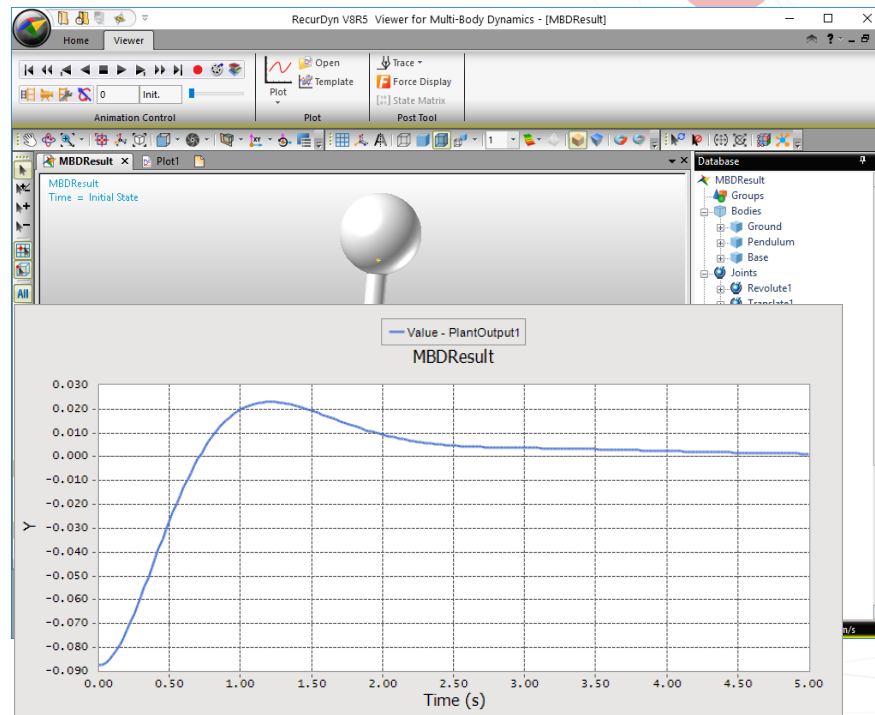
Particleworks

Control

CoLink

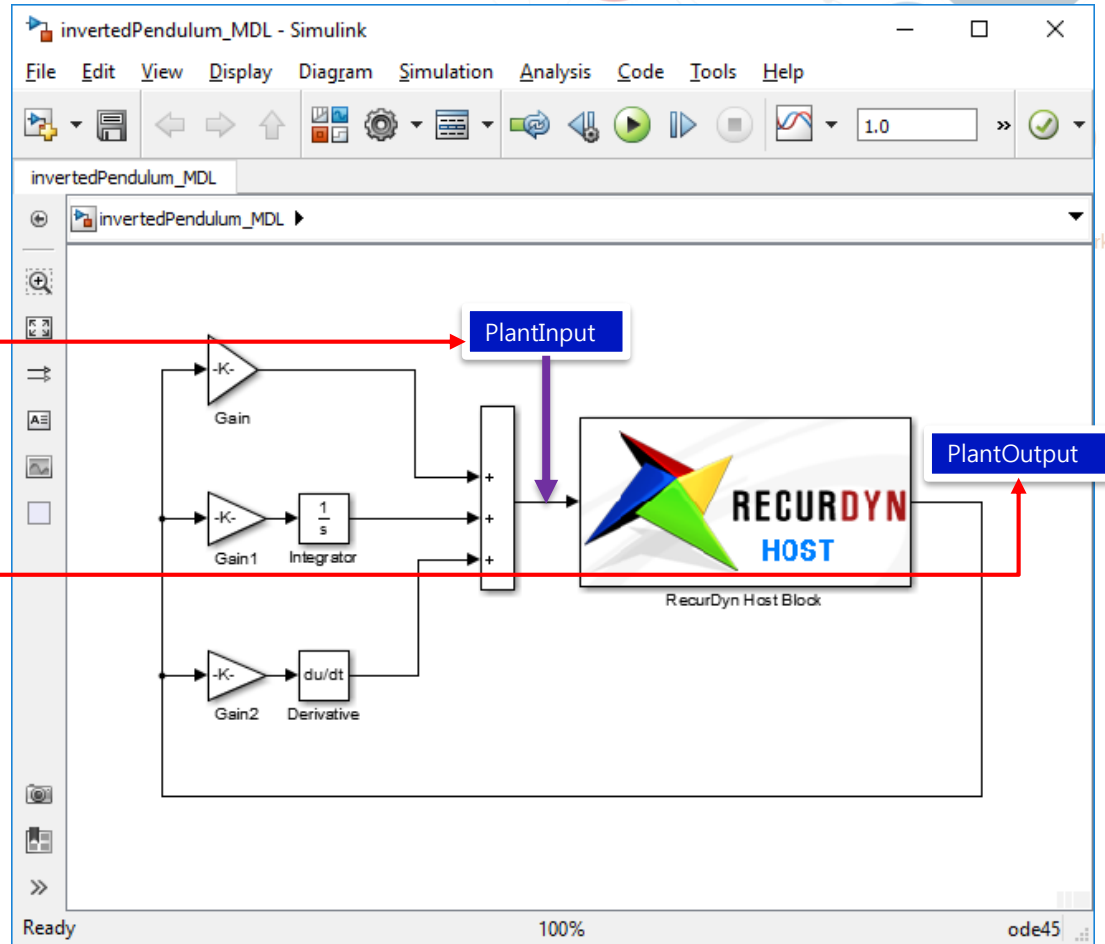
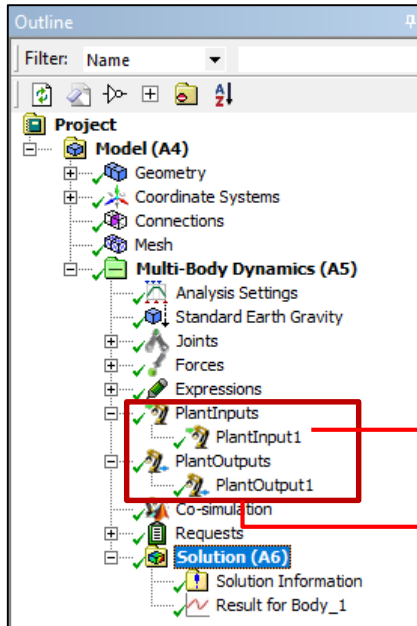
Simulink

1. MBD Post Toolbar 에서 View – **RecurDyn Viewer** 클릭
2. 변경된 결과 확인
 1. 애니메이션
 2. Plot



Play the animation

참고 - 제어 모델 설명



- MBD for ANSYS에서 계산된 Pendulum 의 각도는 PlantOutput 값으로서, Host Block의 output으로서 출력됨
- 이렇게 출력된 각도(PlantOutput)을 이용하여 PID 블록으로 제어된 값은 PlantInput으로서 Host Block 으로 입력됨. (MBD for ANSYS의 PlantInput으로서 입력됨)

MBD for ANSYS Host Block = MBD for ANSYS Model