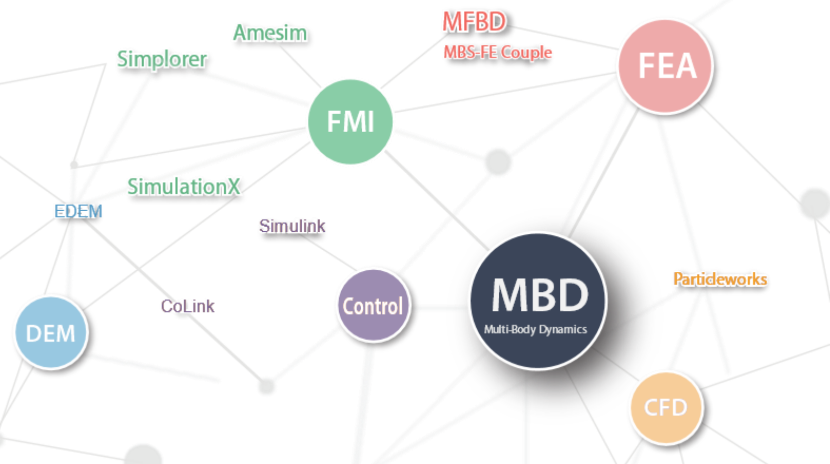
Connect to All

MBD for ANSYS
Multi-Body Dynamics



MBD for ANSYS
Simulink Interface Training

Korean

English

James Kang

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

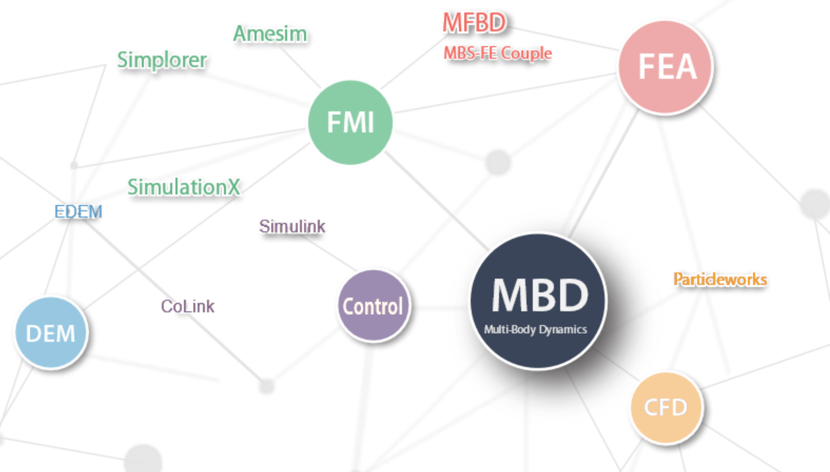
1. Current Directory
2. MBD for ANSYS Host Block
3. Simulink model

❑ Simulation

1. Simulation
2. Post (Animation, Plot)

❑ Improve Simulink Model

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



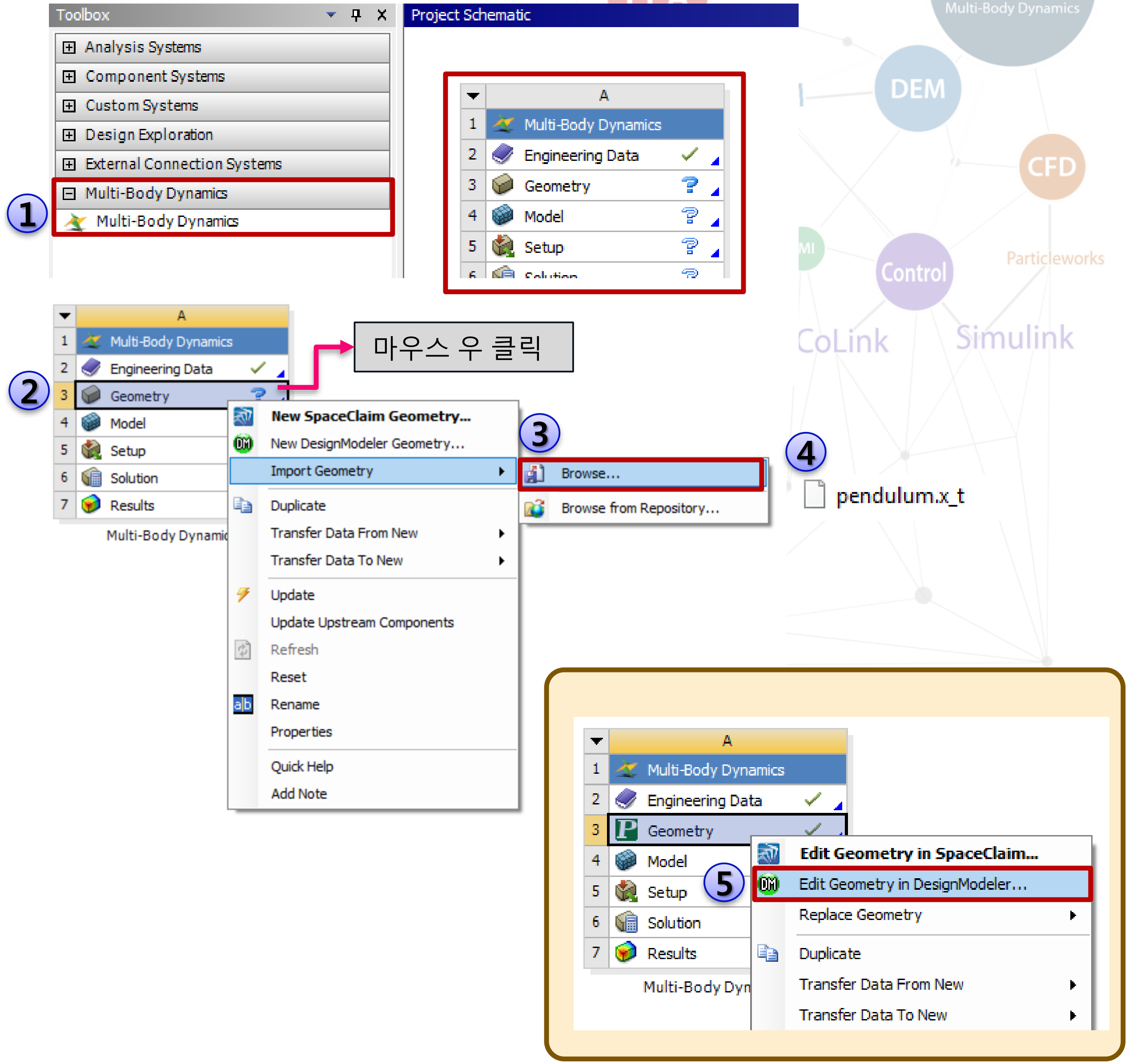
MBD for ANSYS Model

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**

pendulum_CAD-file.zip



Importing Geometry

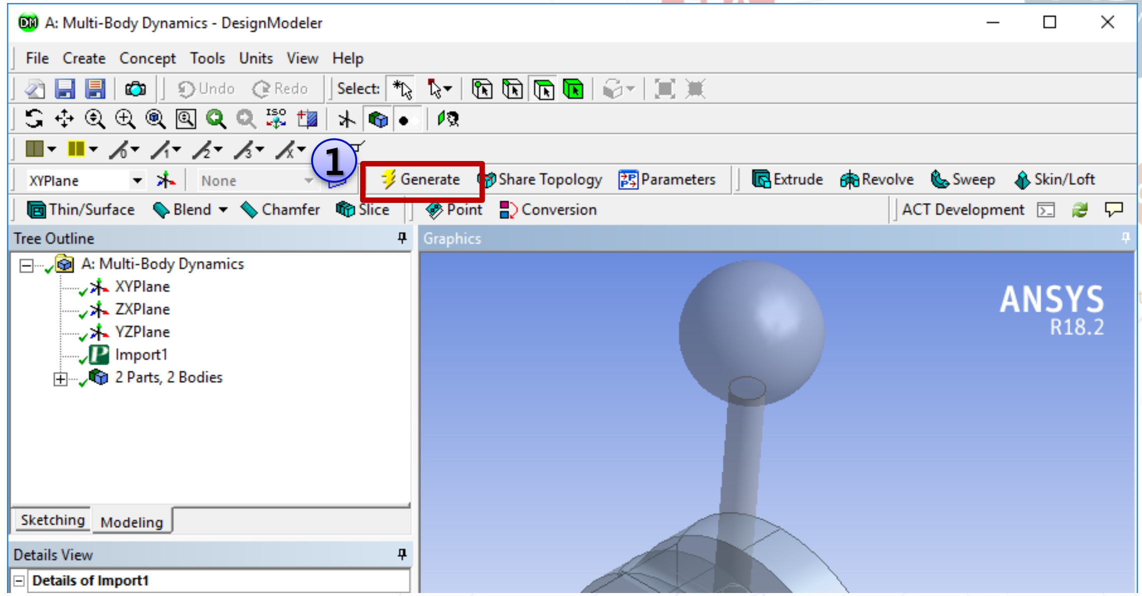
1. Click **Generate** tool in the toolbar of DesignModeler.

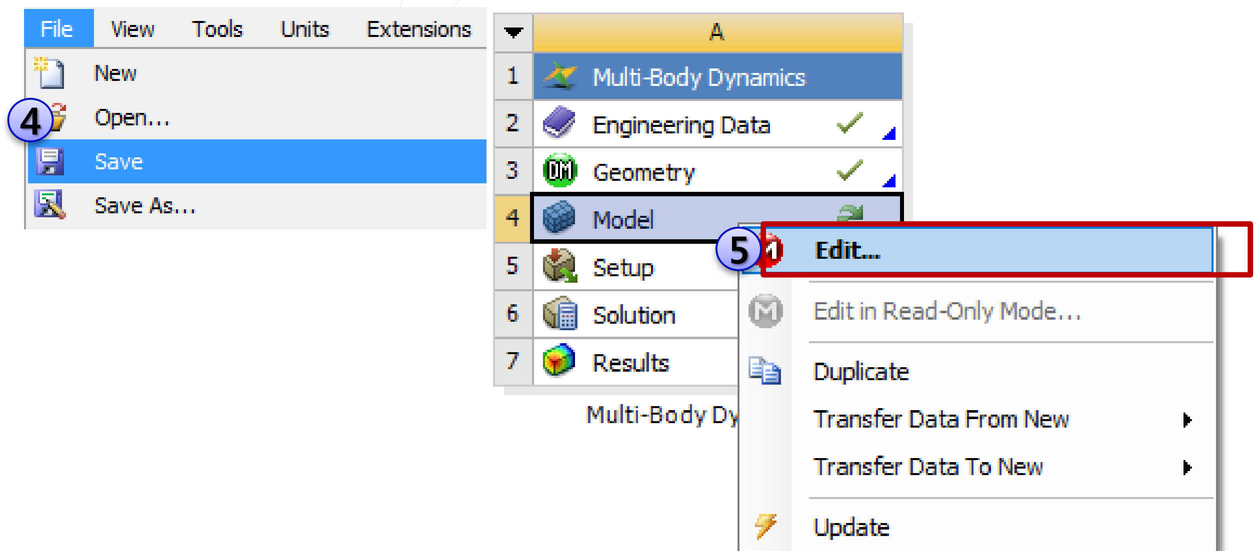
2. You can see the geometry.

3. **Close** DesignModeler

4. **Save** the project

5. **Right mouse button** with the cursor over the **Model** field and select the **Edit...**





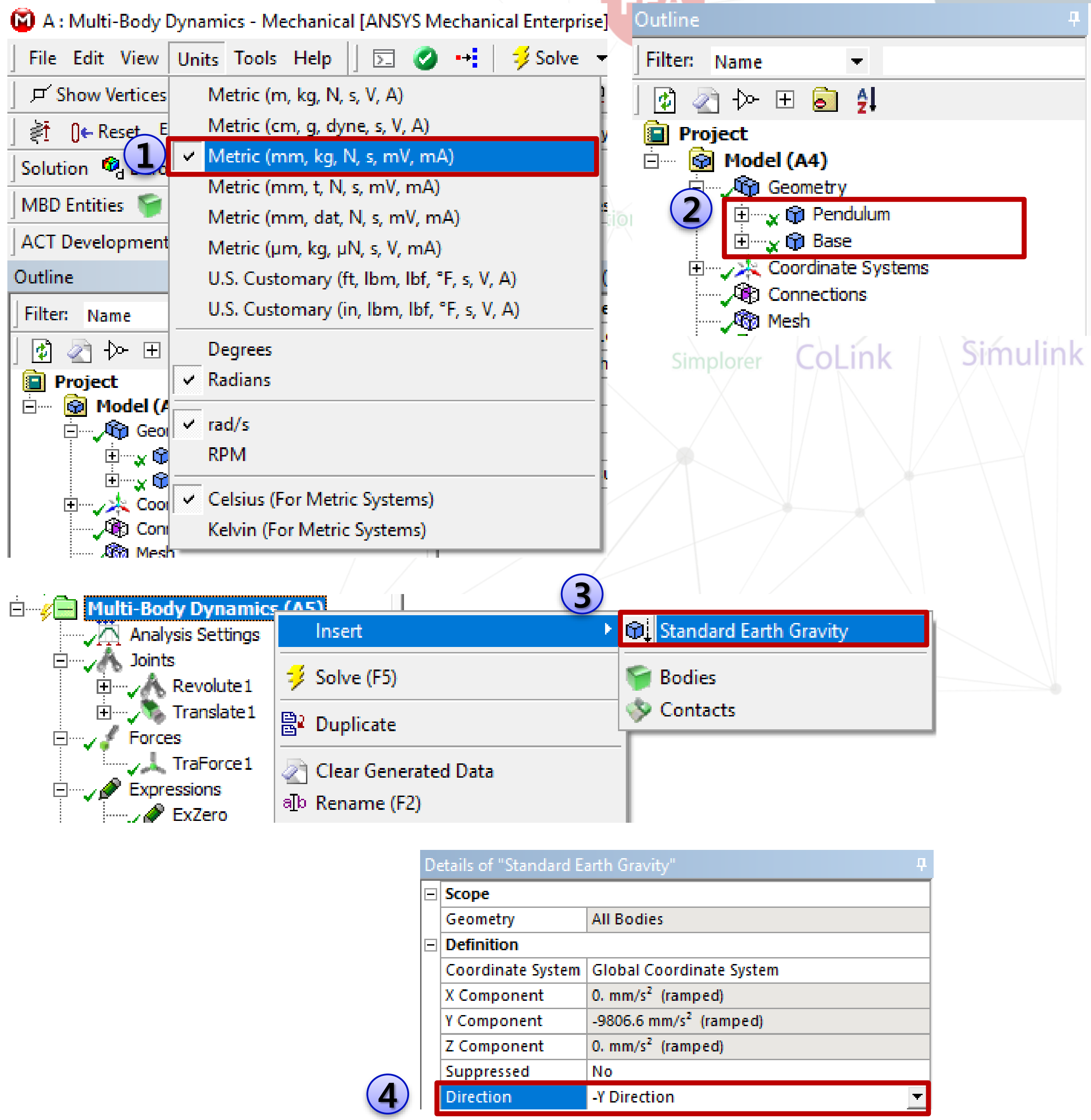
Unit and Gravity

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**

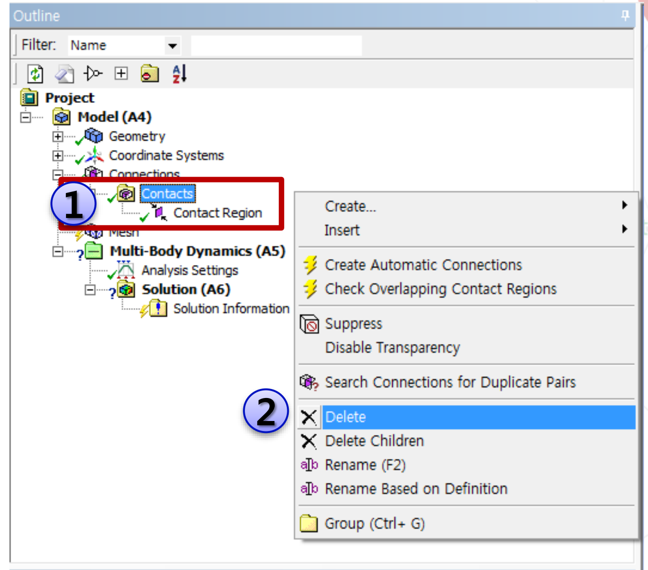


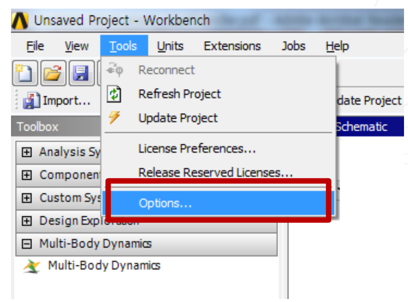
Delete Contact Region

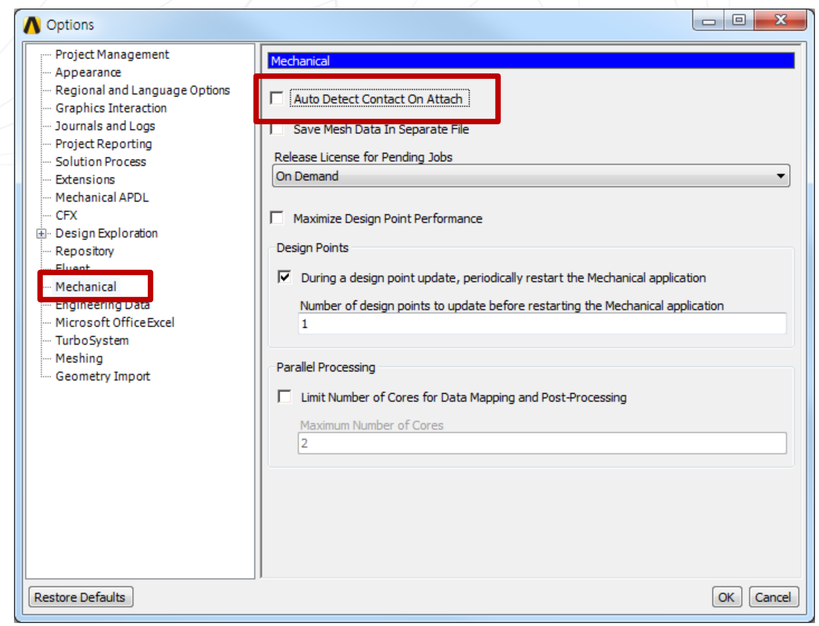
1. Select Contacts in the Outline window.

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)**

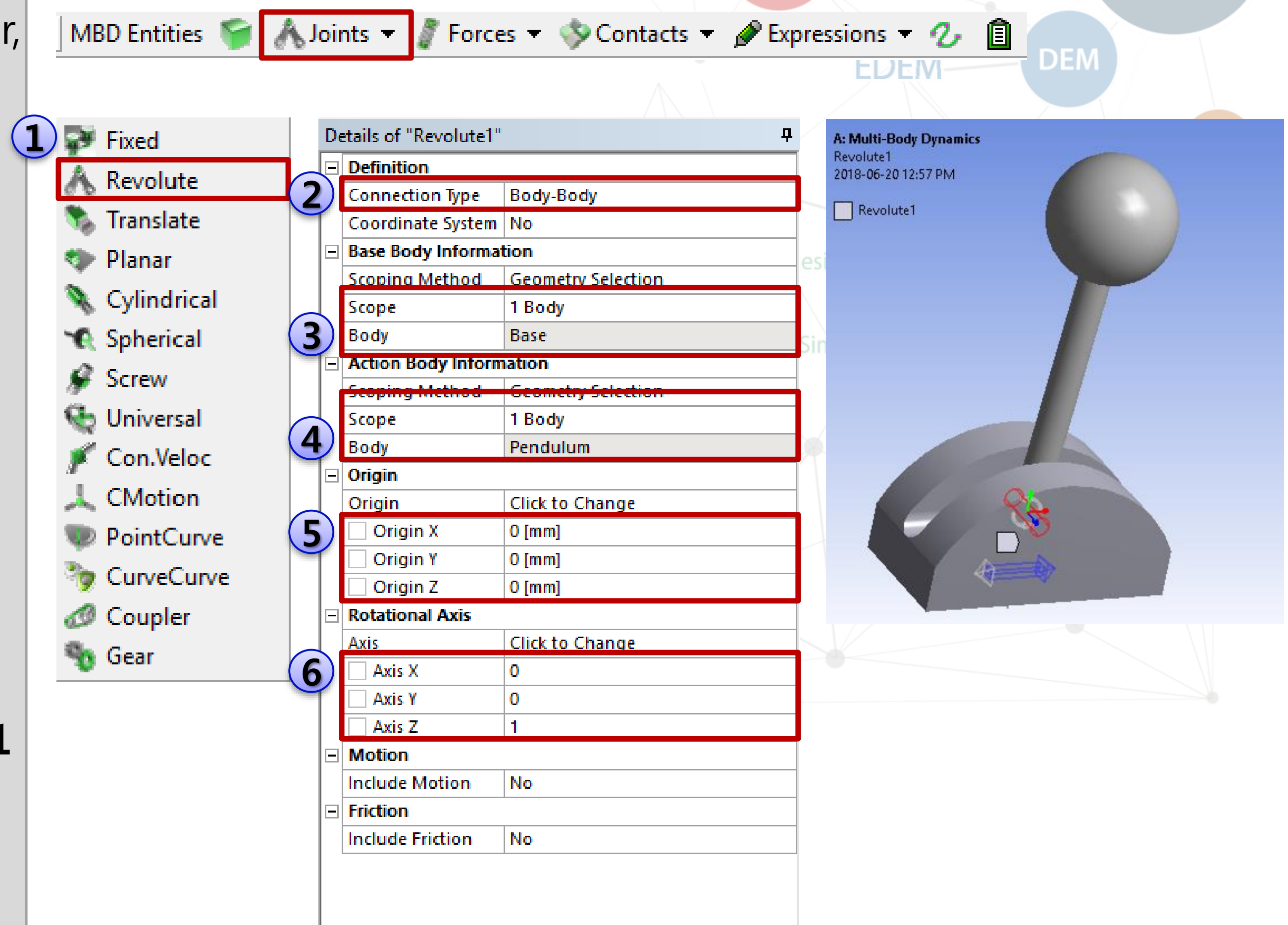






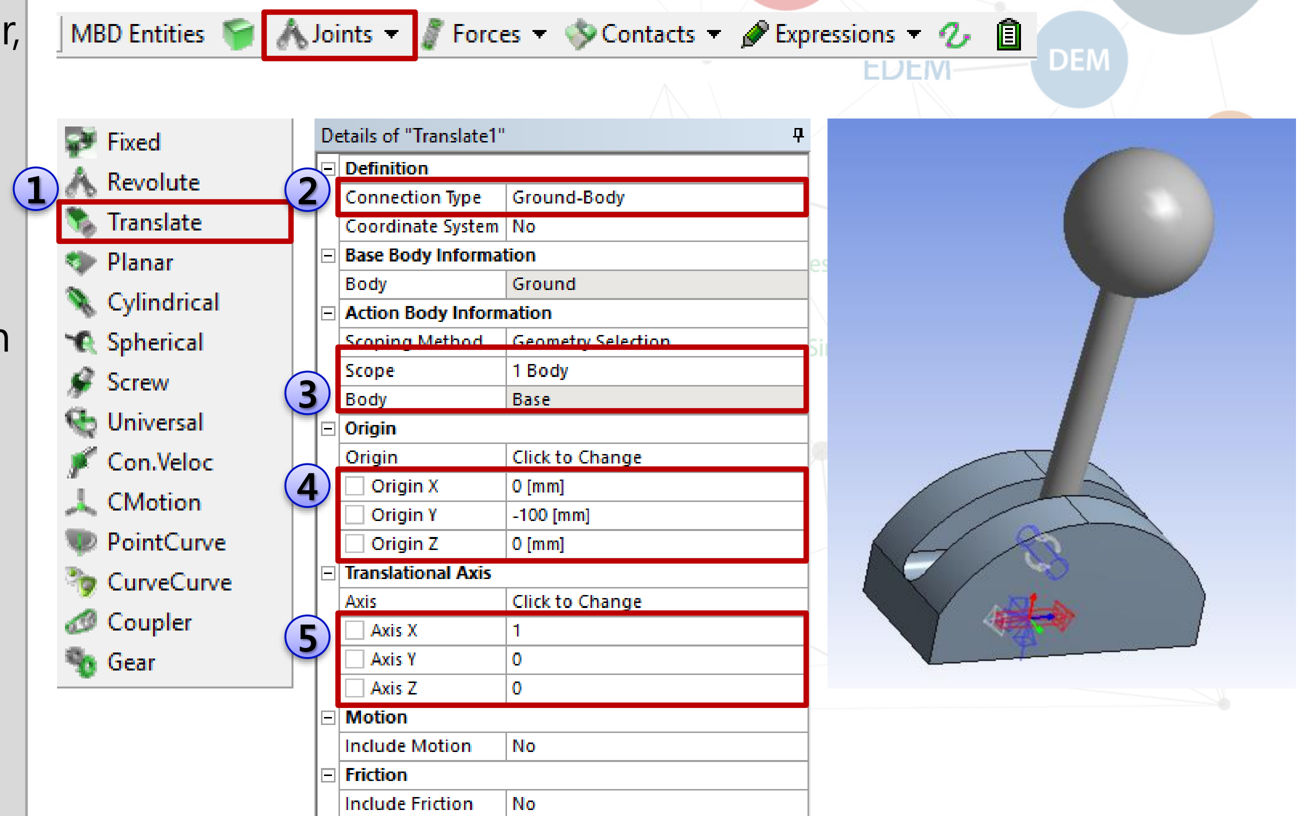
Joint – Revolute Joint

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**



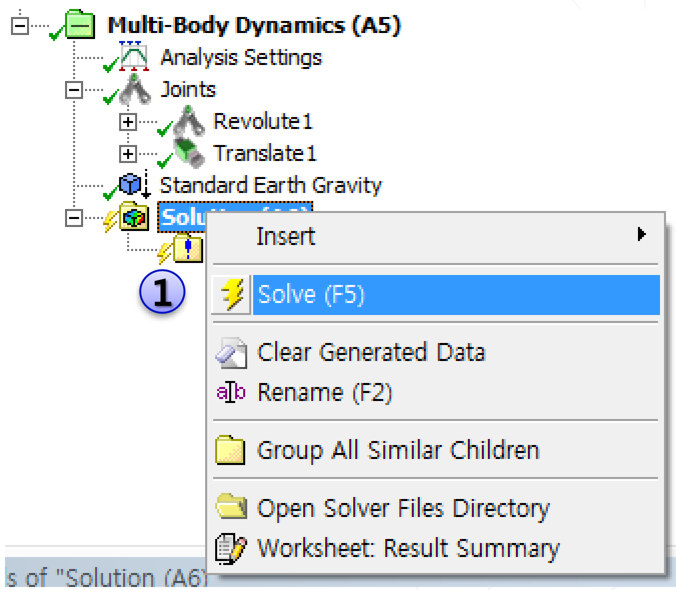
Joint – Translational Joint

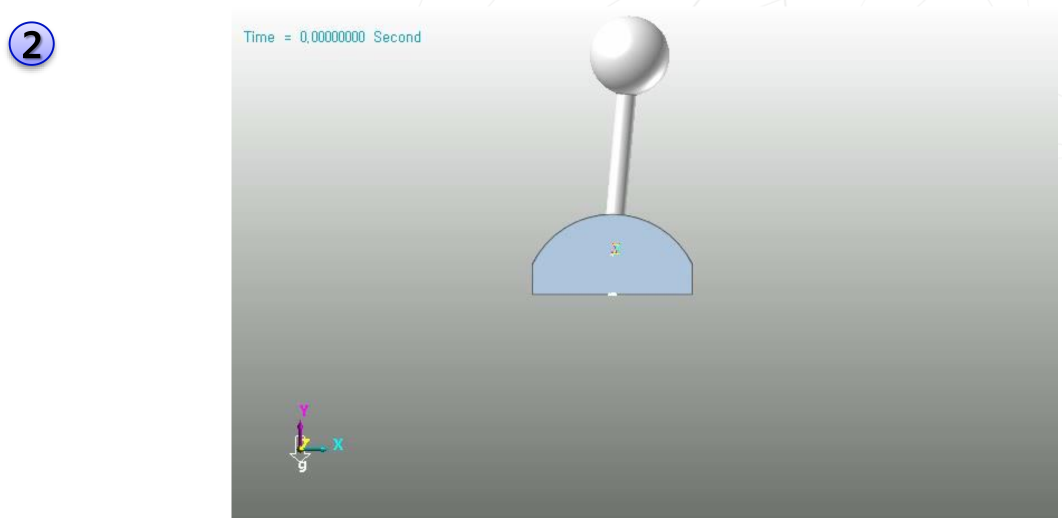
1. In the MBD Entities Toolbar, select **Joint** – **Translate**
2. Set **Connection Type** to **Ground-Body**
3. Select ‘**Base**’ for the Action Body
4. Set Origin to **0,-100,0**
5. Set Rotational Axis **1,0,0**



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.

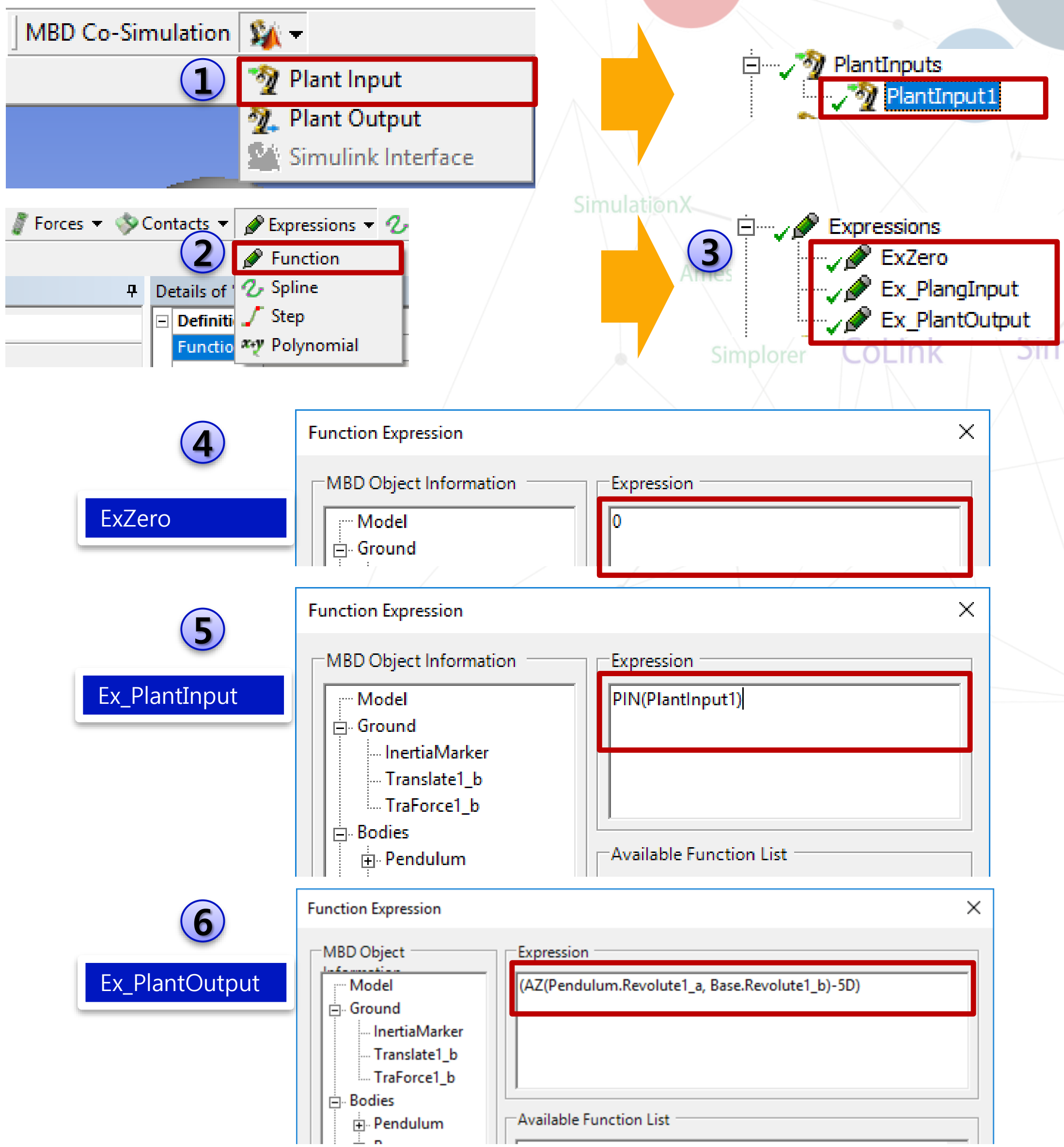




Play the animation

Plant Input & Plant Output

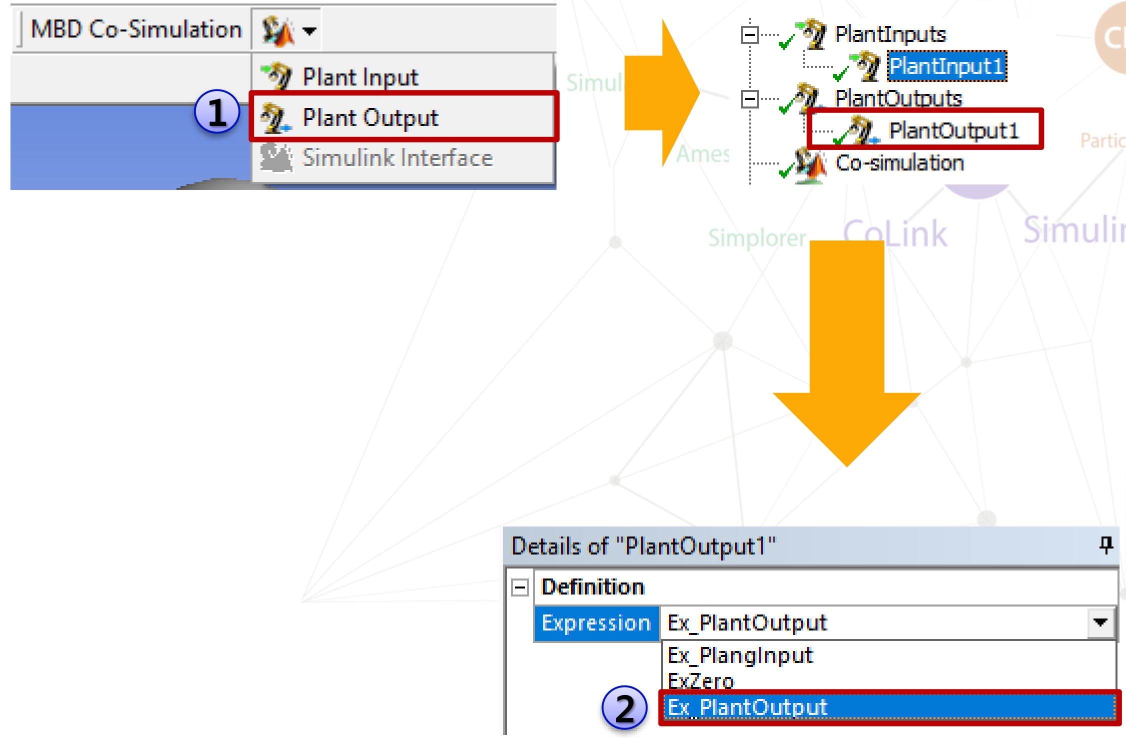
1. In the MBD Co-Simulation Toolbar, select **Plant Input**
2. In the MBD Entities Toolbar, select **Expressions – Function**
3. **Rename** the name of the function Expression to **ExZero** and **create 2 more function expressions** and **rename** them to **Ex_PlantInput**, **Ex_PlantOutput**
4. Input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



Plant Input & Plant Output

1. In the MBD Co-Simulation Toolbar, select **Plant Output**

2. Select **Ex_PlantOutput** for **the expression**



Force – Translational Force

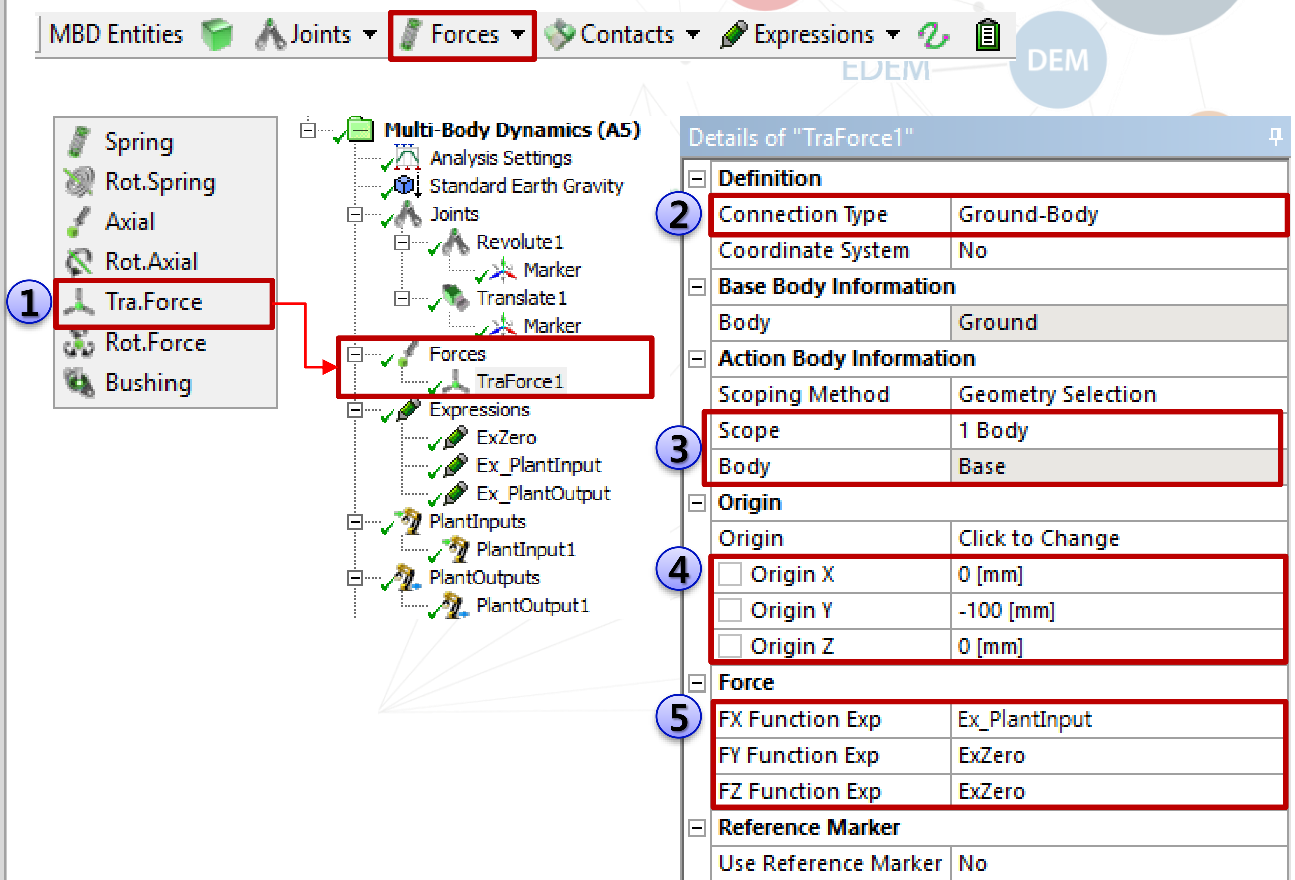
1. In the MBD Entities Toolbar, select Force – **Tra.Force**

2. Set **Connection Type** to **Ground-Body**

3. Select '**Base**' for the Base Body

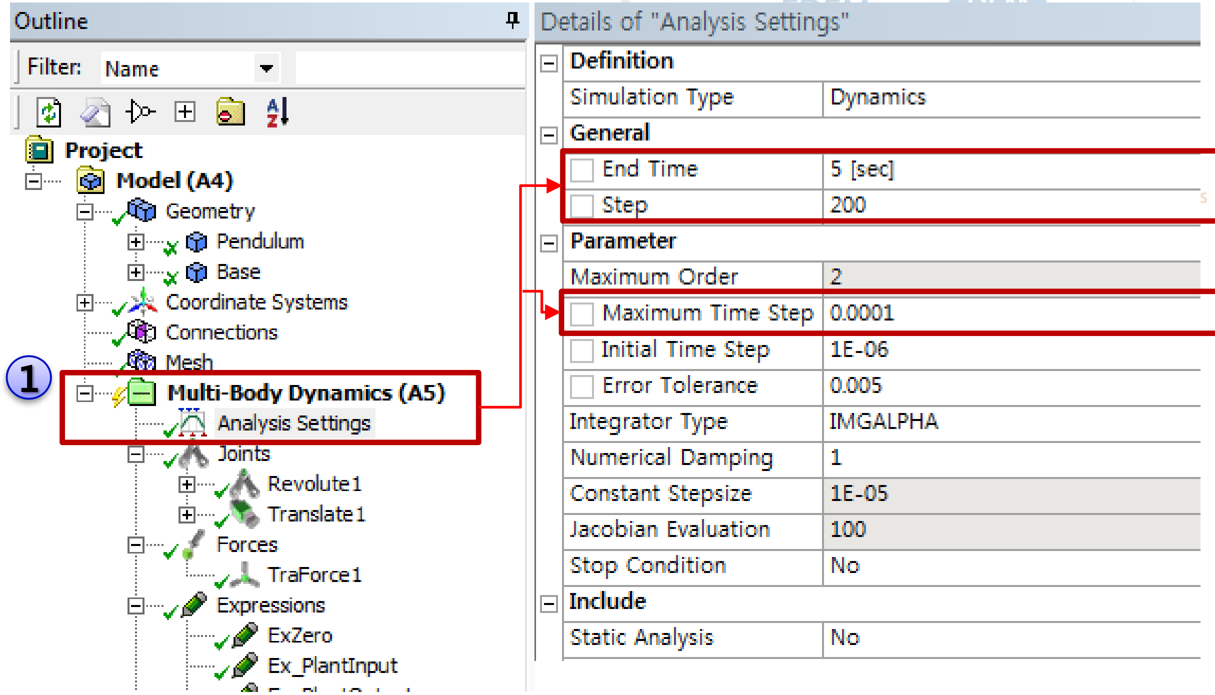
4. Set Origin **0,-100,0**

5. Select **Ex_PlantInput** for **FX Function Exp**, and select **ExZero** for **FY Function Exp** and **FZ Function Exp**



Analysis Settings

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



Create Simulink Interface

1. In the MBD Co-Simulation Toolbar, select **Simulink Interface**

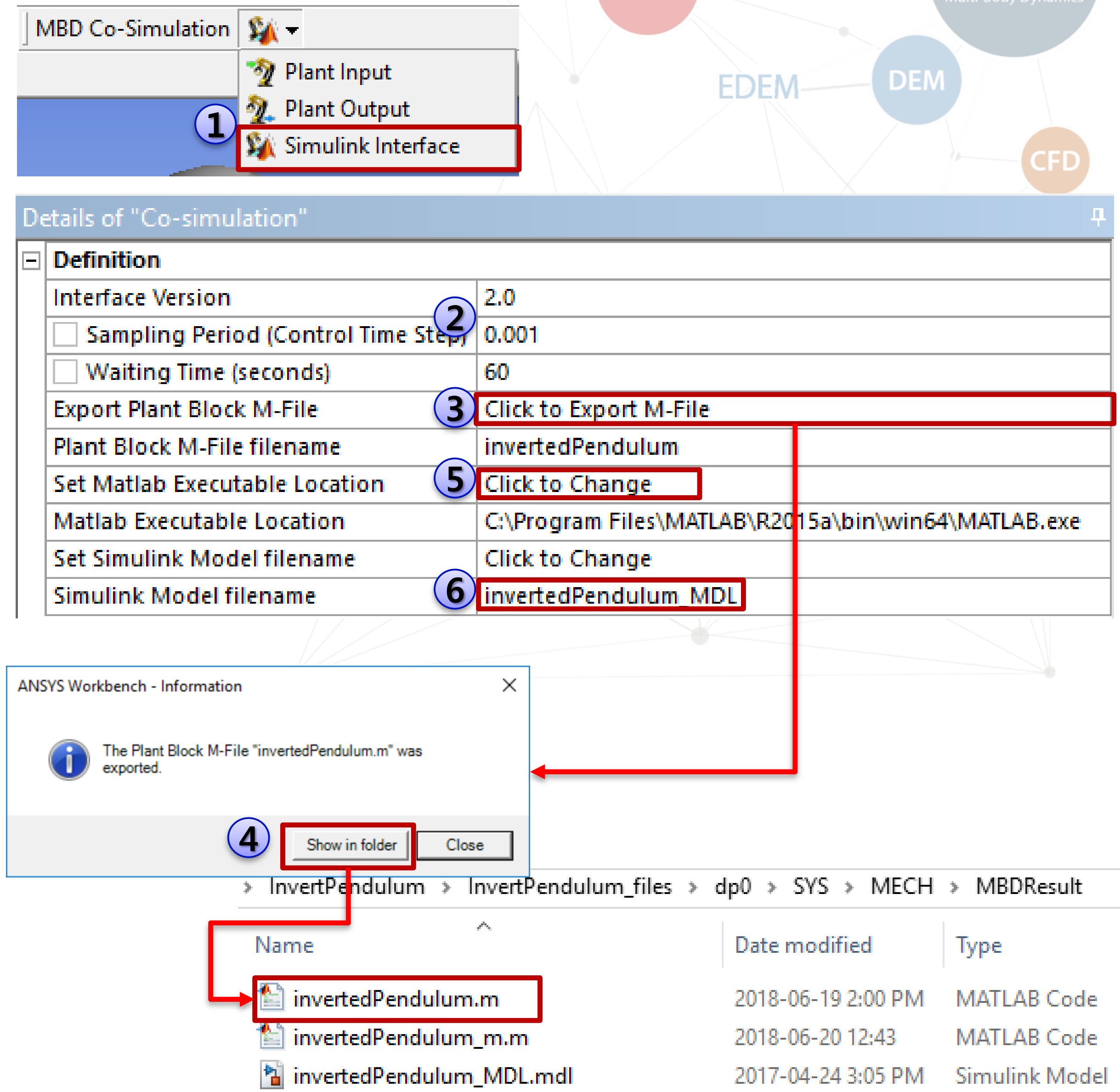
2. Set as below:
   Interface Version = **2.0**
   Sampling Period = **0.001**
   (Set Interface Version = 3.0,
   if you use MATLAB 2016a or later)

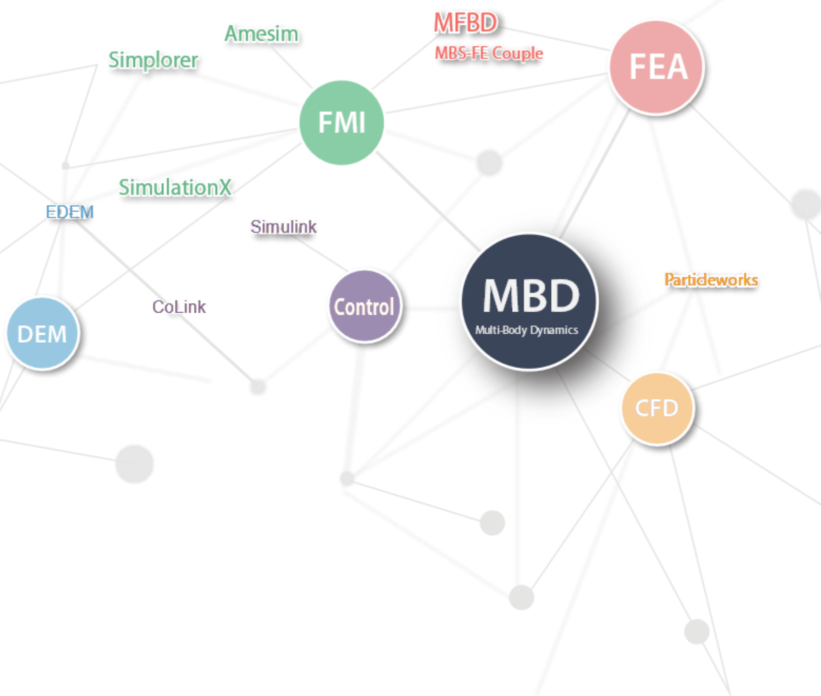
3. Set Plant Block M-File filename to **invertedPendulum** and select '**Click to Export M-File**'

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

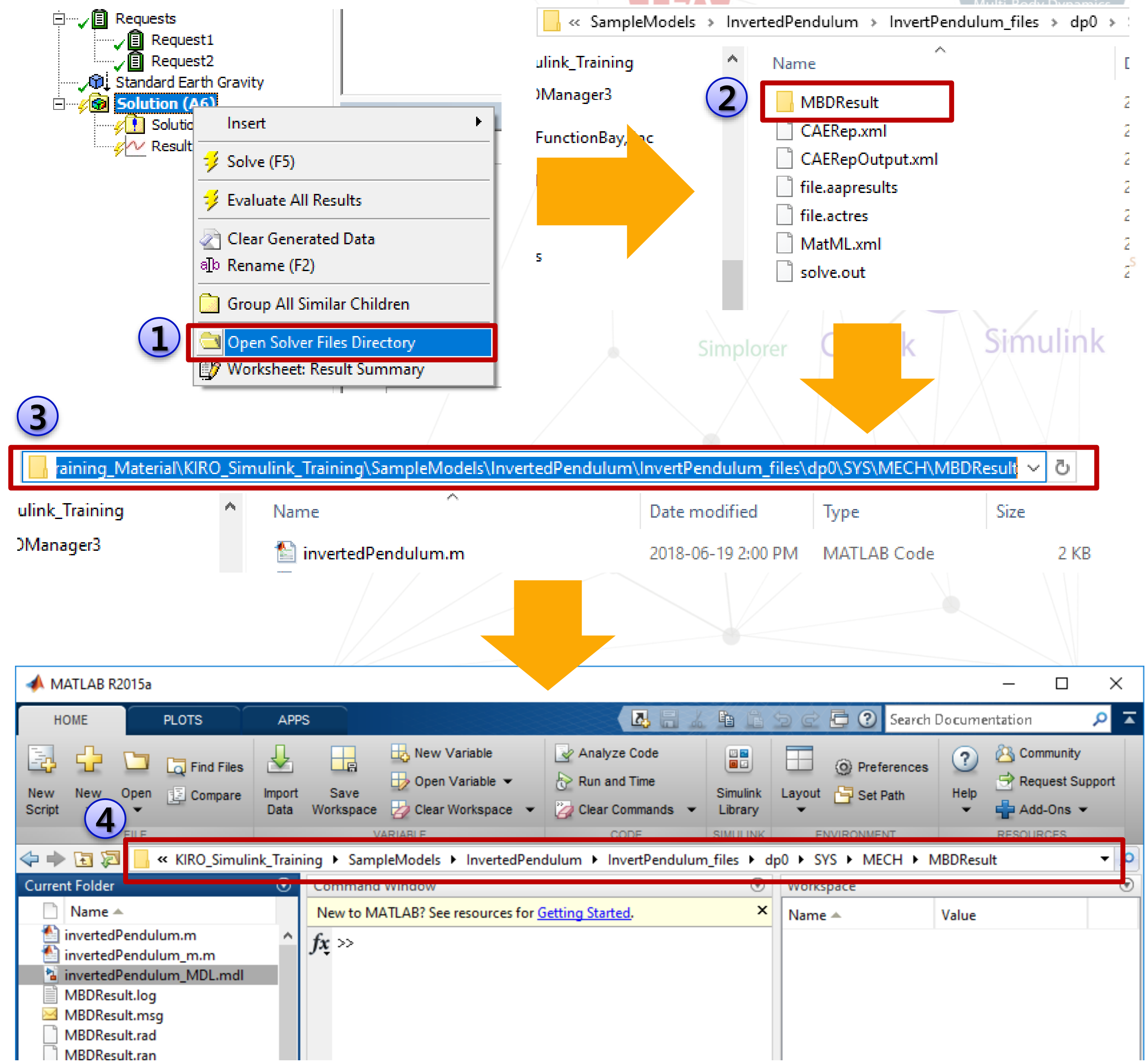
Working directory

1. Select **Open Solver Files Directory**

2. Select **MBDResult** folder

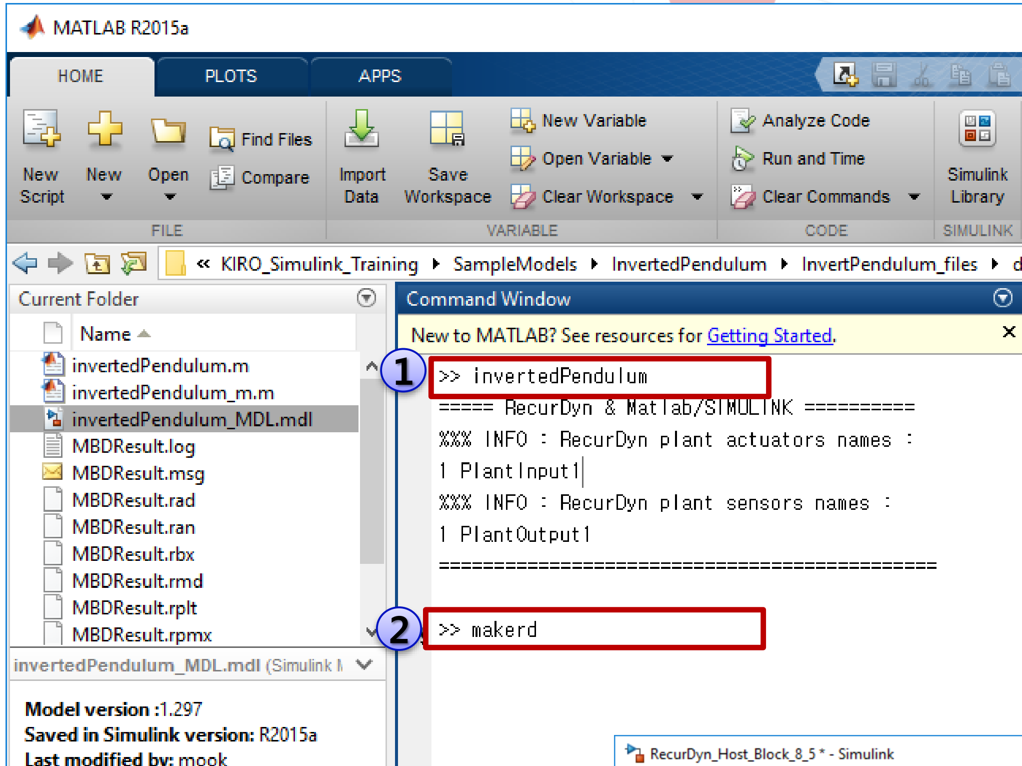
3. **Copy** the **full path** where **invertedPendulum.m** is located

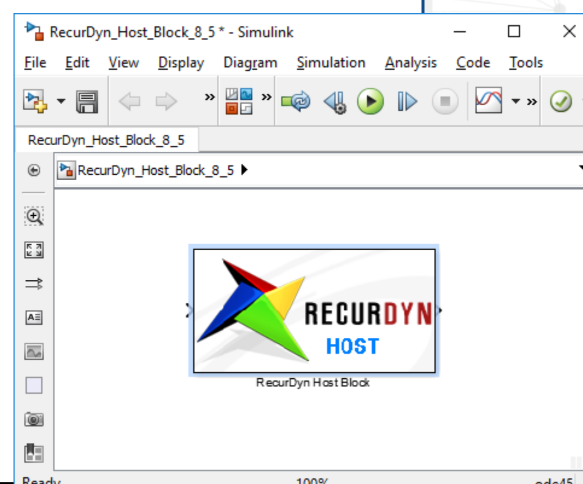
4. Execute **Matlab** and **paste** the path to **Current Folder** of Matlab.



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

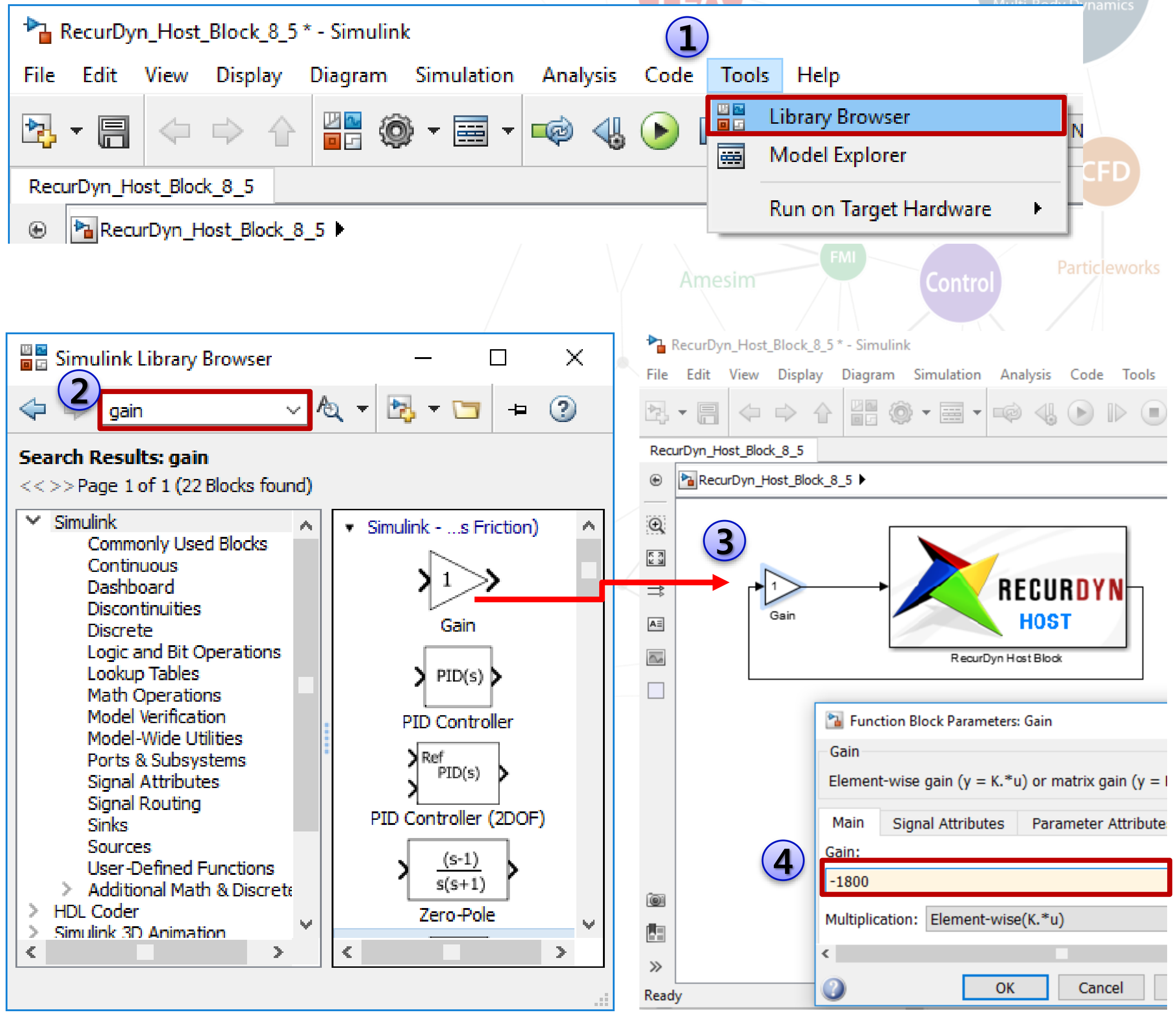
Simulink Model - Simulink Block

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**

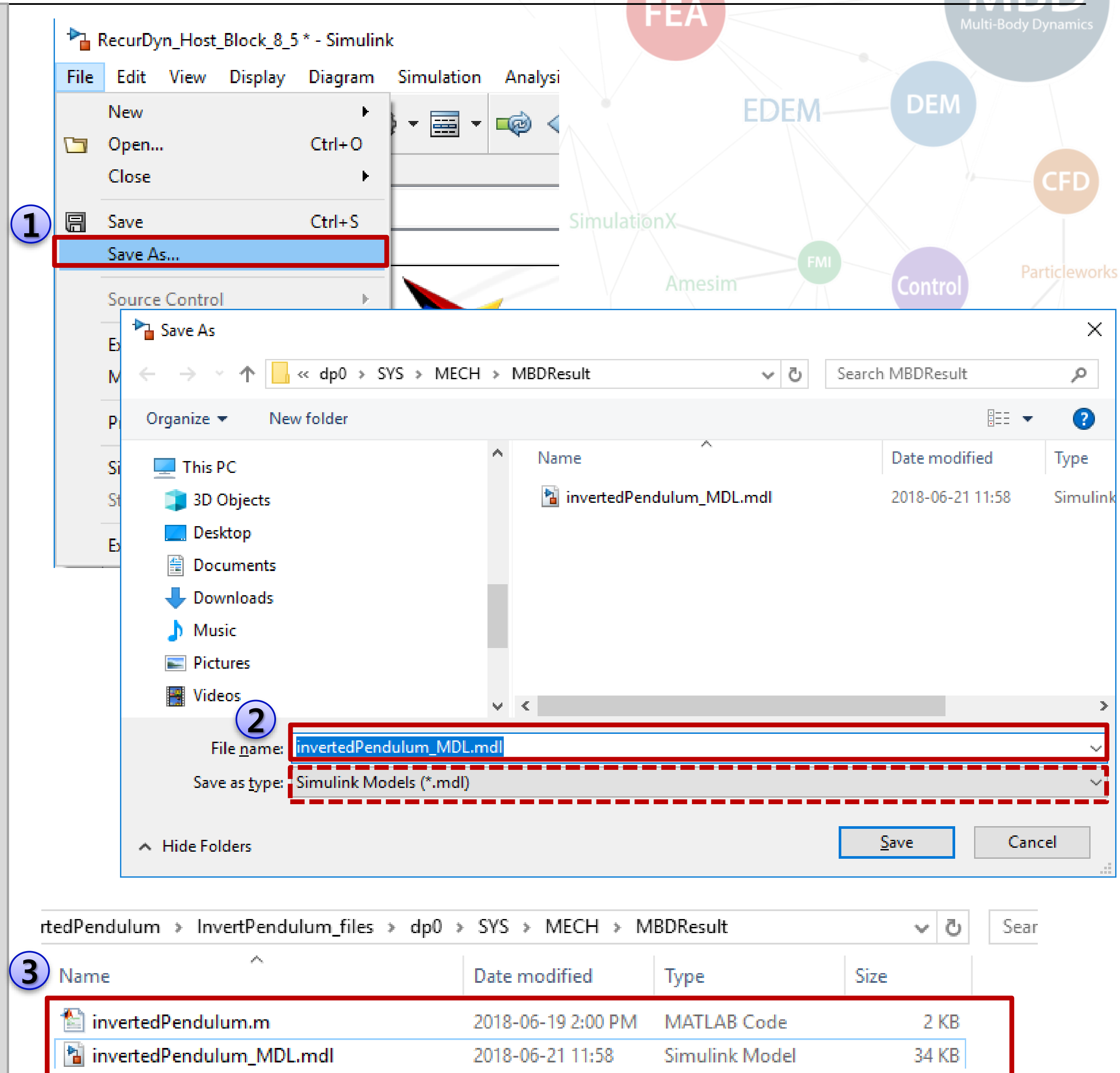


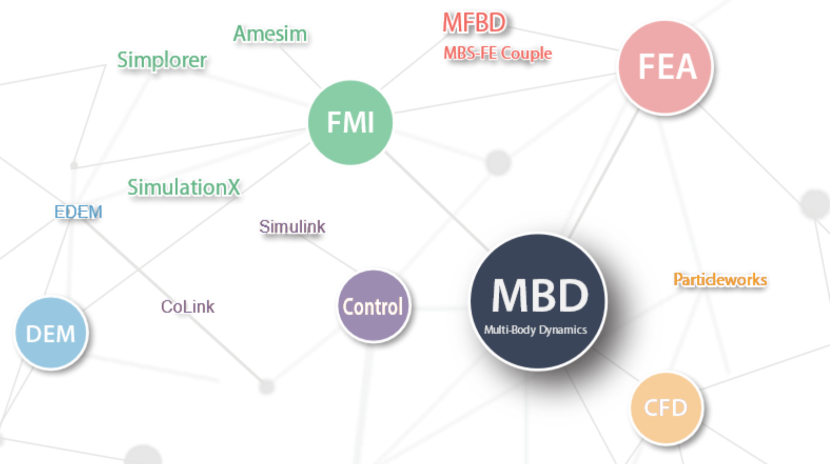
Simulink Model - save

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



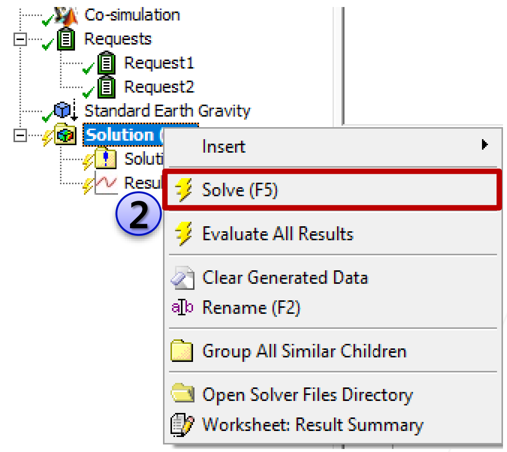


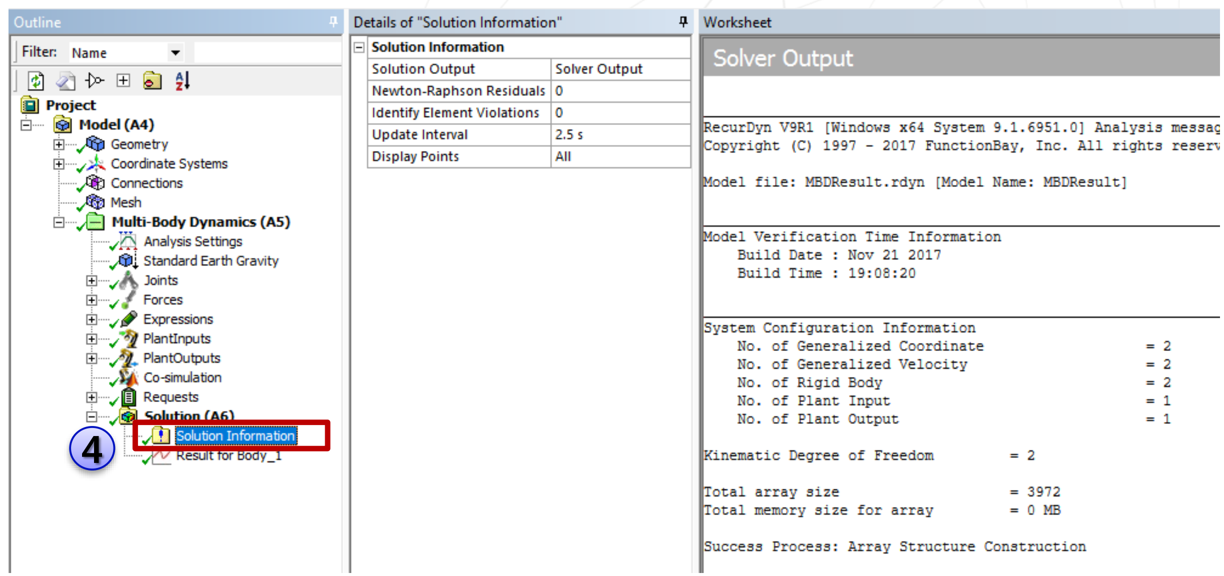
Simulation

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.

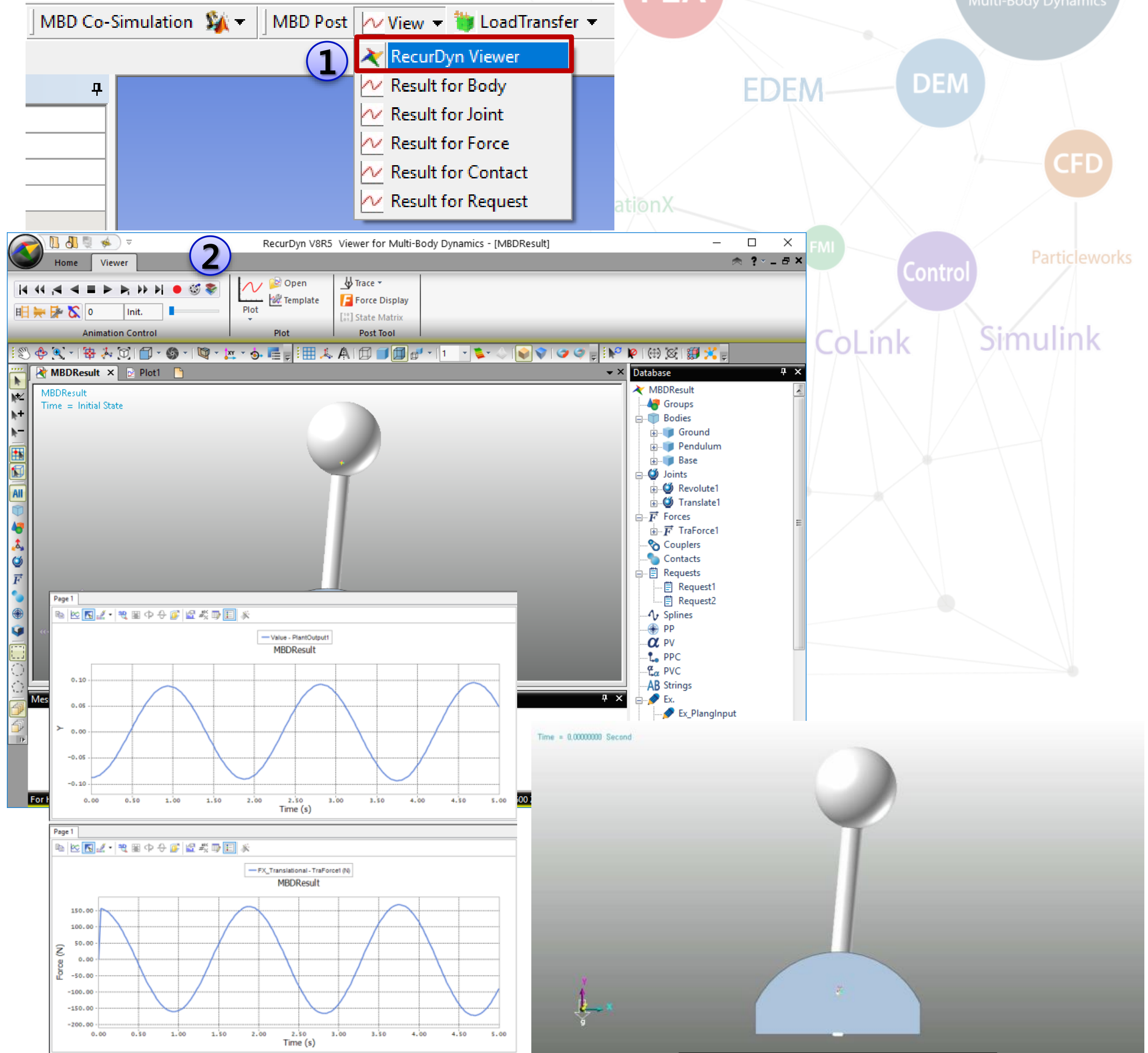




Post

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]

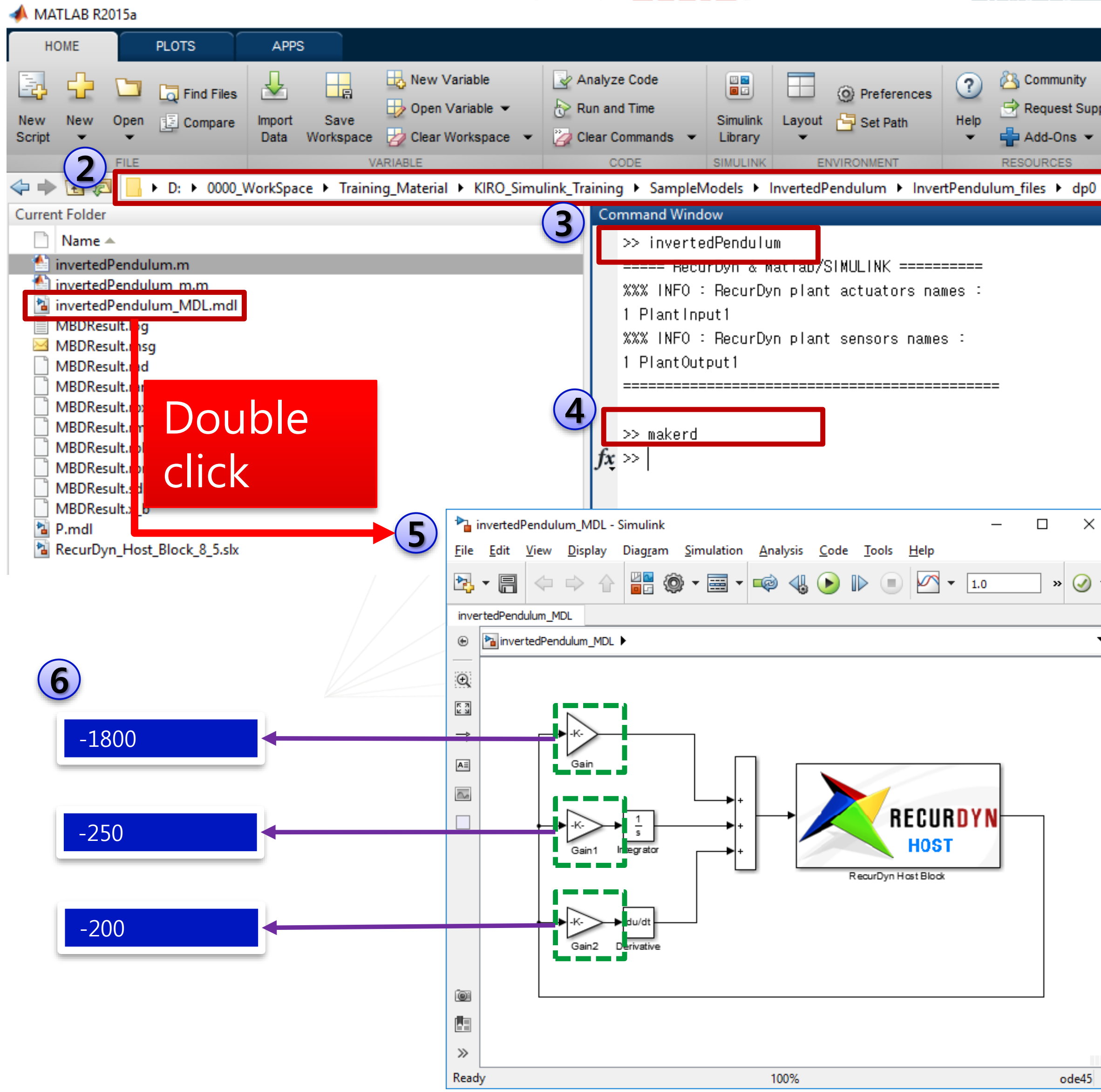
4. Input **makerd** in the **command window** + [Enter]

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

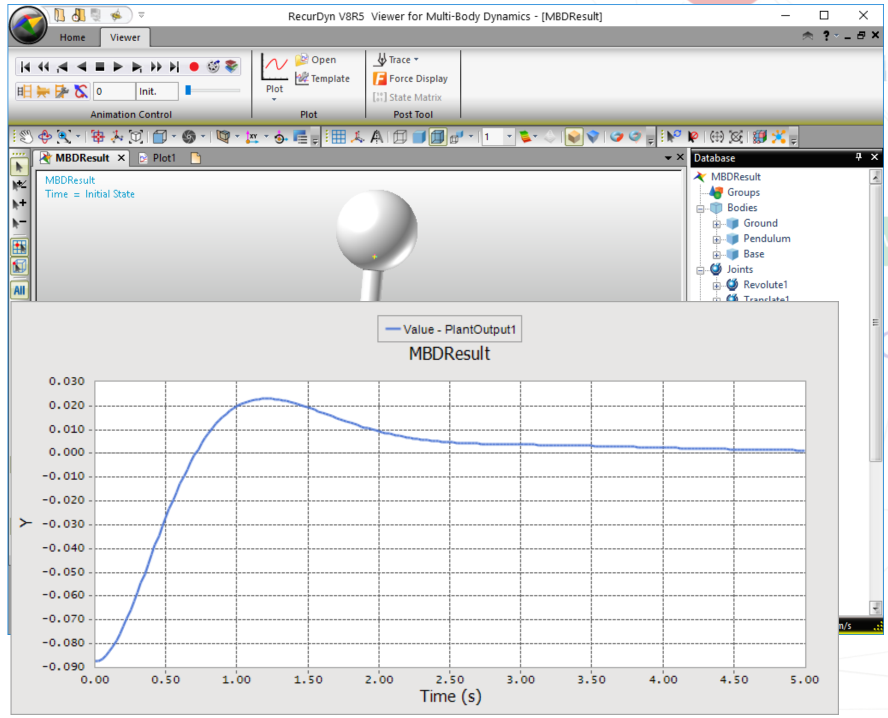
8. **Solve** the model from MBD for ANSYS

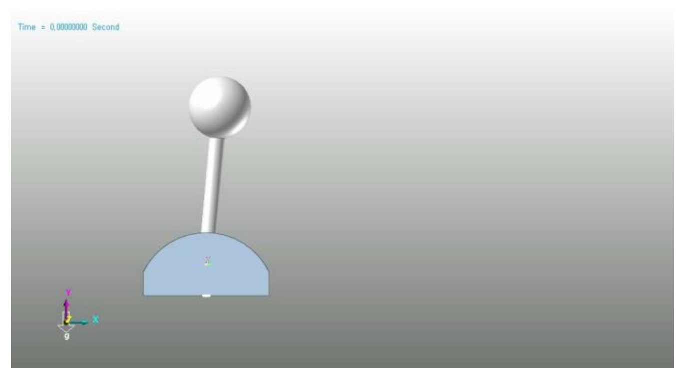


Post

1. In the MBD Post Toolbar, select View – **RecurDyn Viewer**

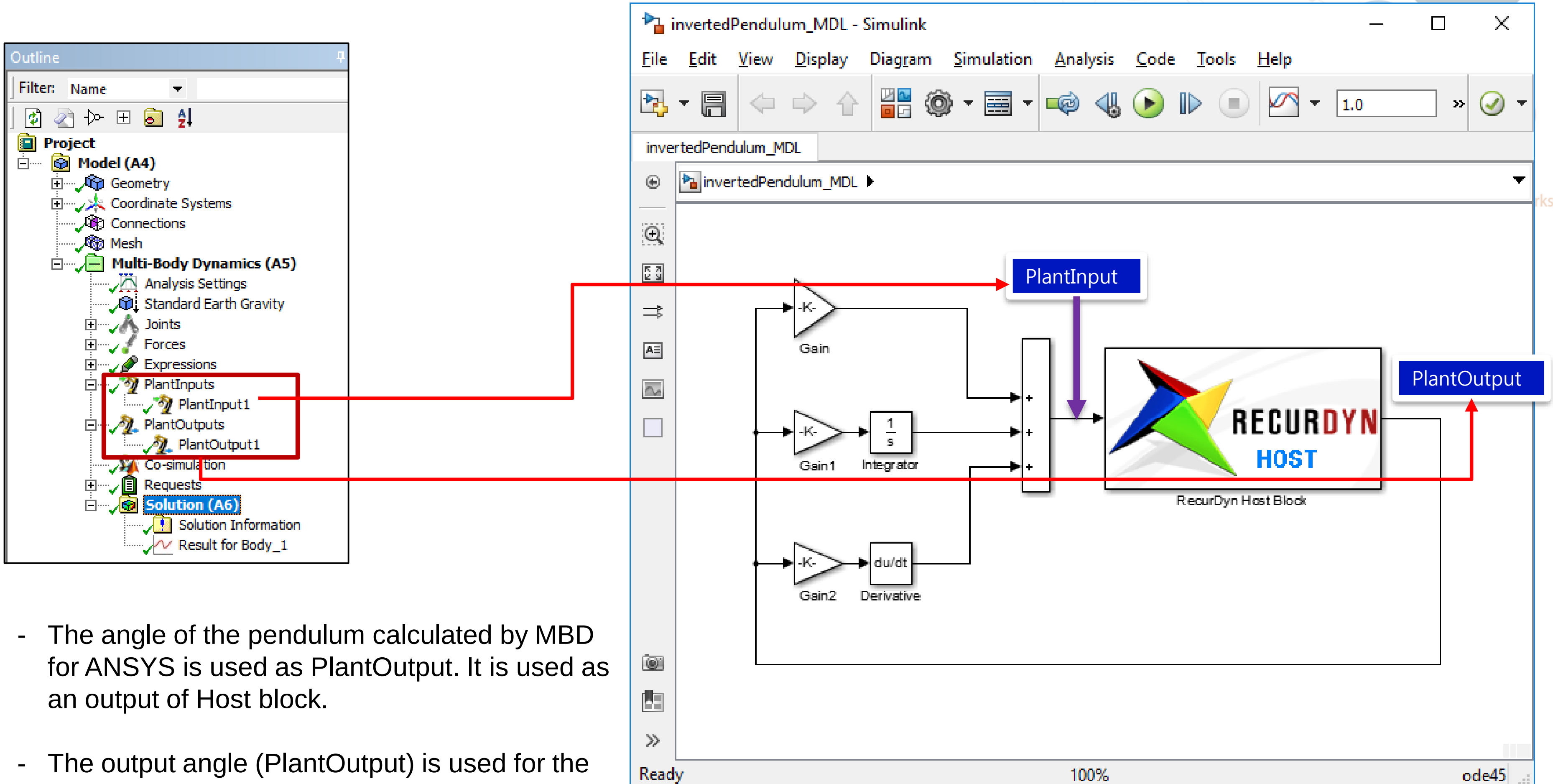
2. Review the results in RecurDyn Viewer
   1) Animation
   2) Plot





Play the animation

Explanation about the PID control model



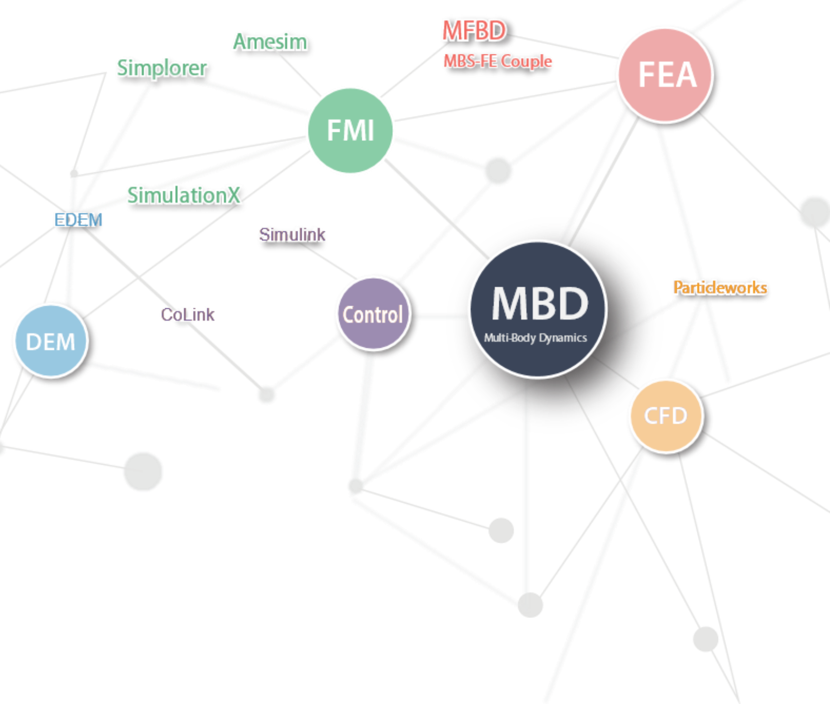
- The angle of the pendulum calculated by MBD for ANSYS is used as PlantOutput. It is used as an output of Host block.

- 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

Korean

English

강재묵

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

Index

❑ 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

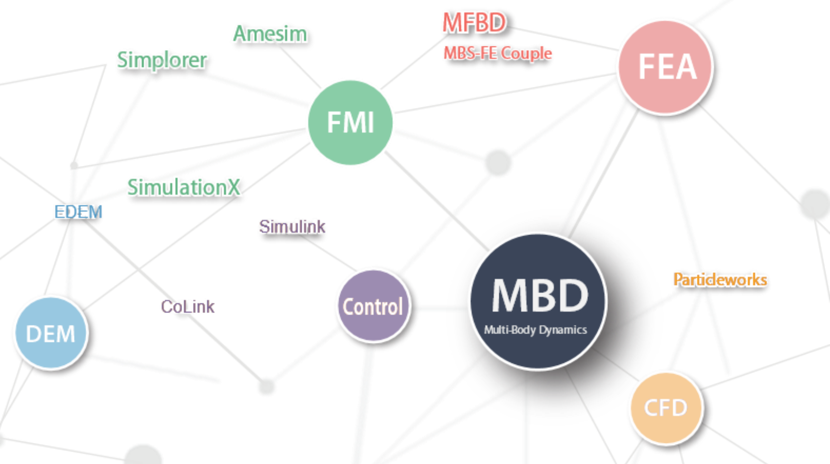
1. Current Directory 설정
2. RecurDyn Host Block 생성
3. Simulink 모델링

❑ Simulation

1. Simulation
2. Post (Animation, Plot)

❑ Simulink Model Upgrade

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



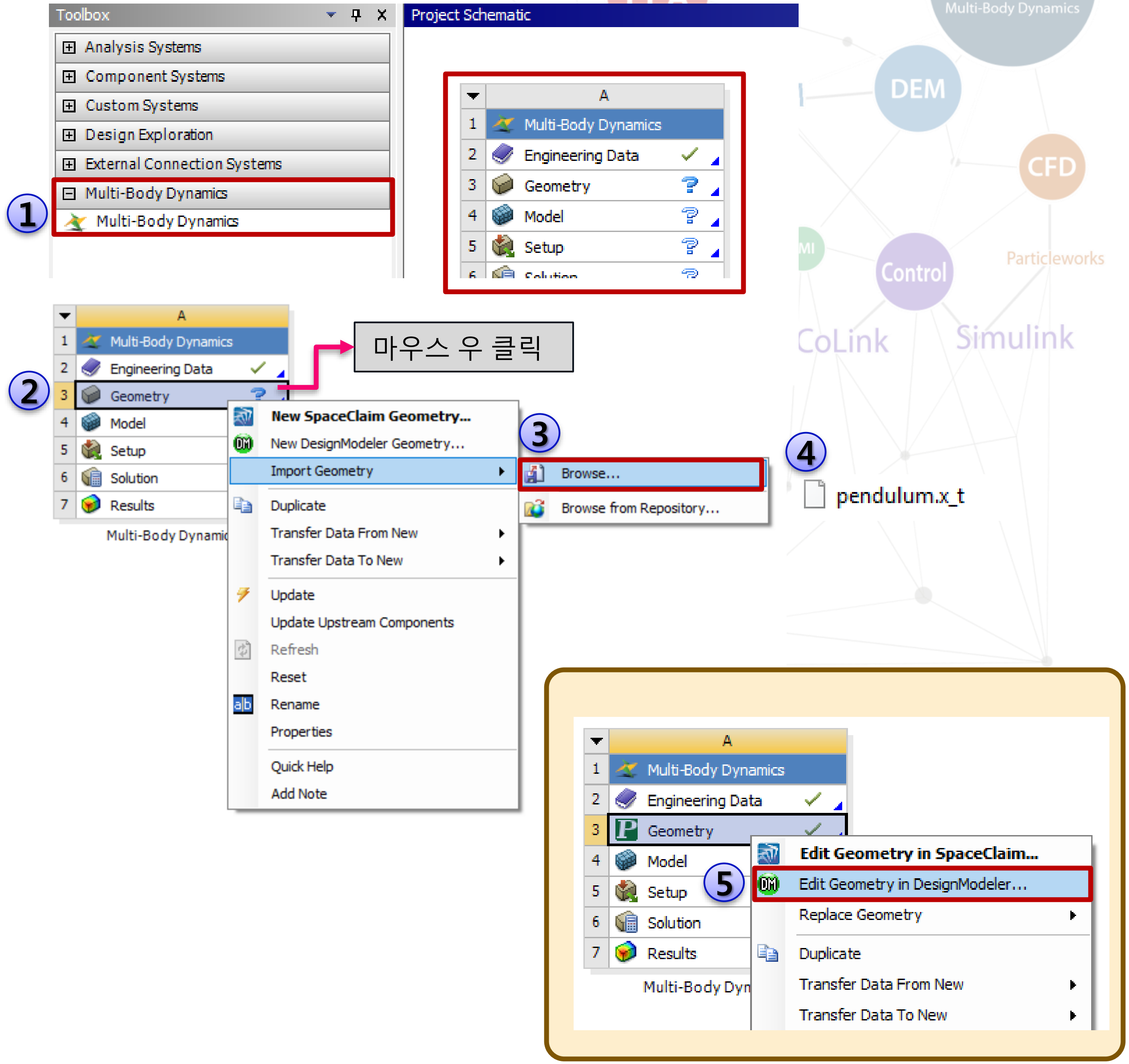
MBD for ANSYS 모델링

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

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

pendulum_CAD-file.zip



Import CAD file

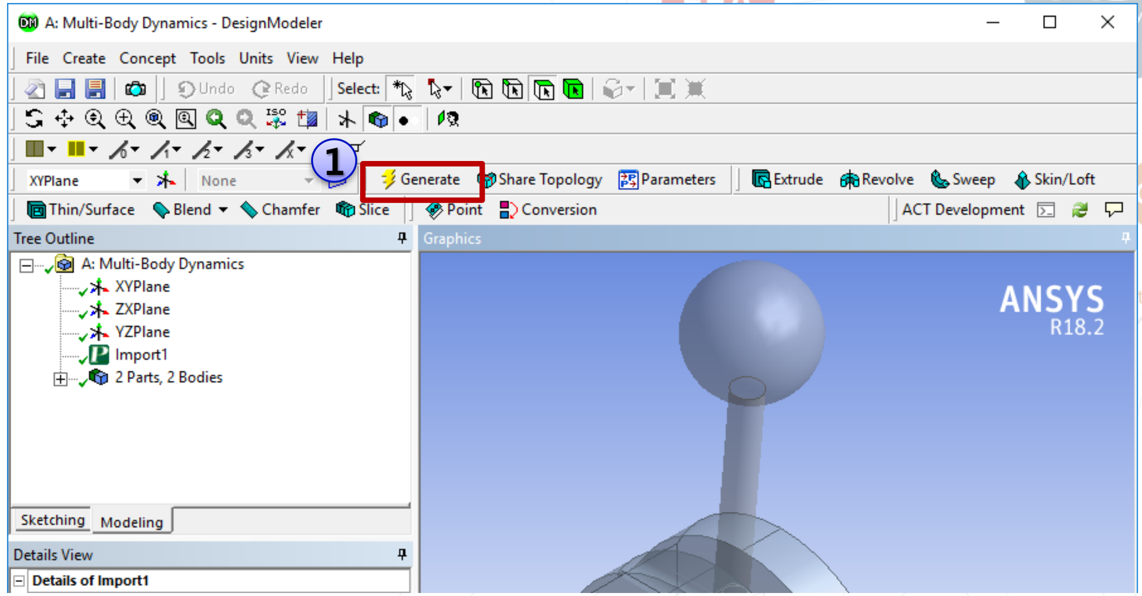
1. DesignModeler 앱이 실행되면 **Generate** 메뉴를 클릭 합니다.

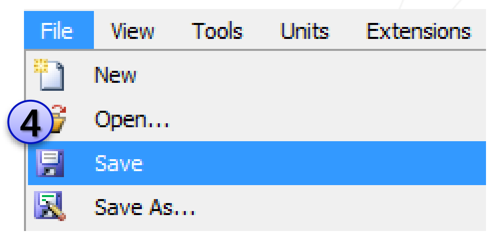
2. 생성된 Geometry 를 확인 합니다.

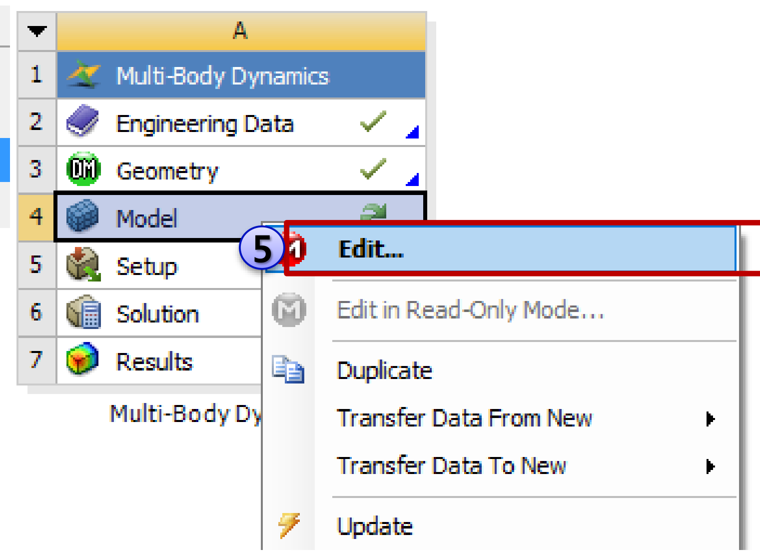
3. DesignModeler 앱을 **종료** 합니다.

4. 프로젝트를 저장(**Save**)합니다.

5. Project Schematic에서 **Model** 셀의 마우스 **우클릭** 메뉴에서 **Edit**를 누릅니다.







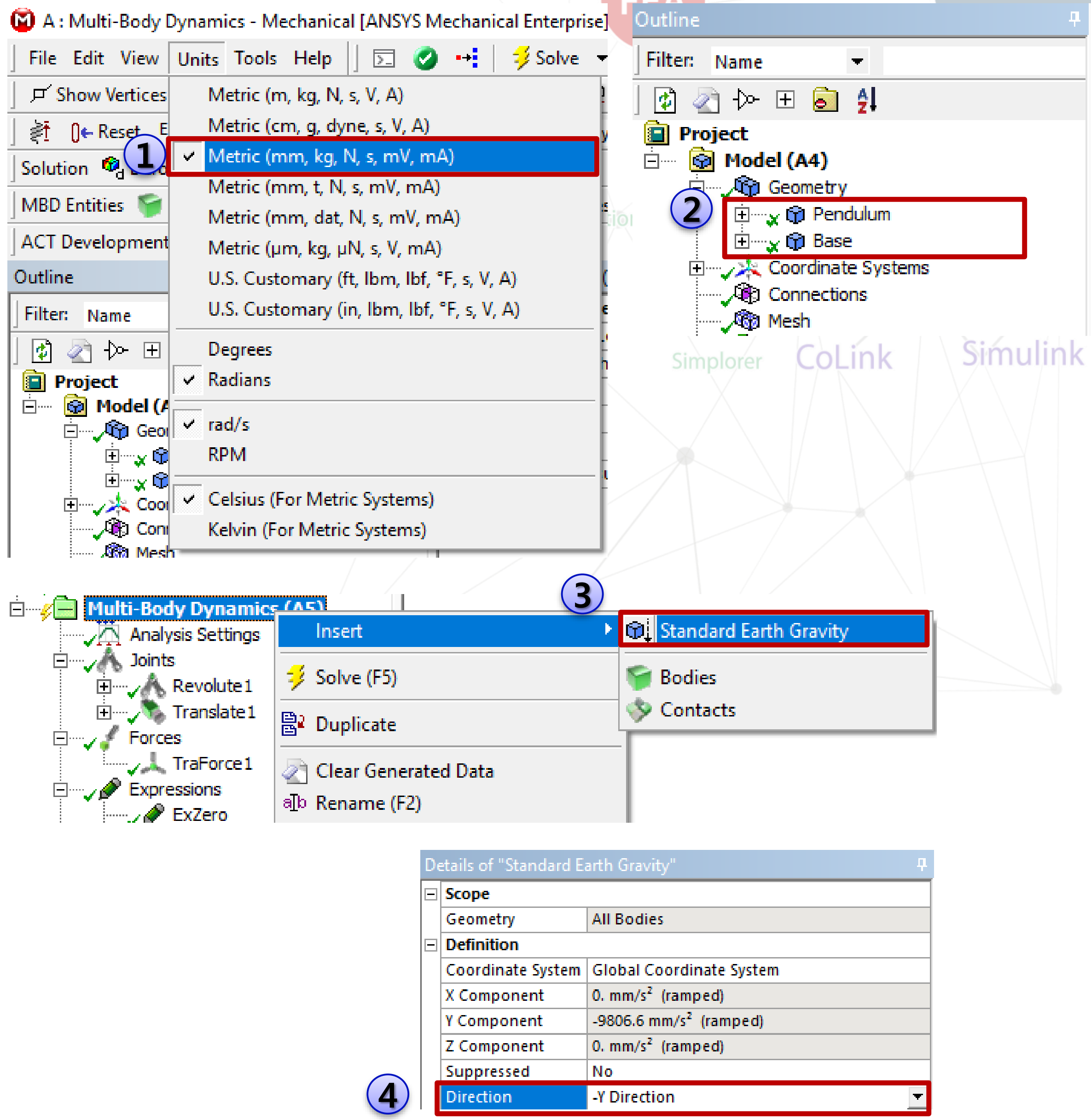
Set unit, change geometry names, creation gravity

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

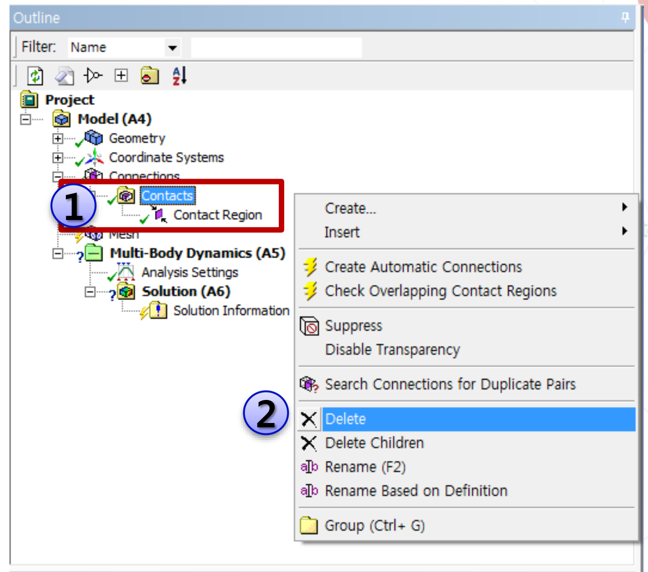


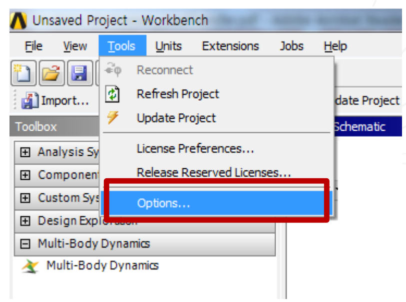
Delete Contact Region

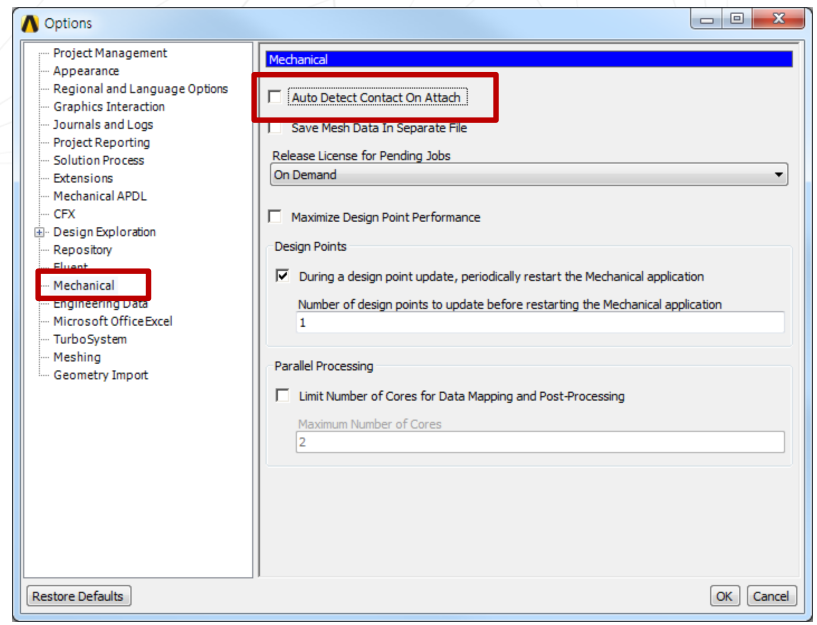
1. Outline 윈도우에서 Contacts 선택

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

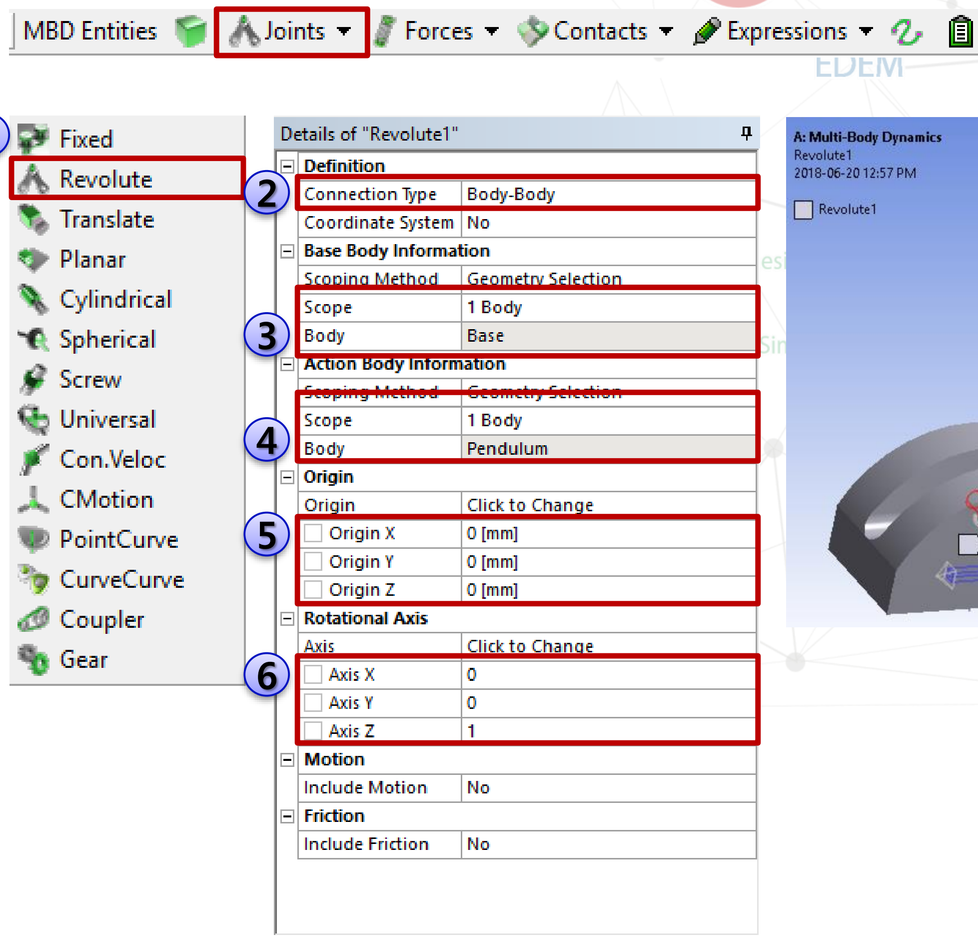
2. Detail 창에서 Connection Type 을 **Body-Body** 로 변경

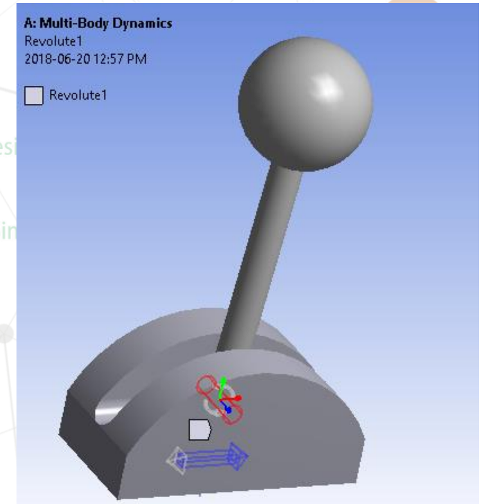
3. Base Body 로 '**Base**' Geometry 선택

4. Action Body 로 '**Pendulum**' Geometry 선택

5. Origin 은 **0,0,0** 으로 설정

6. Rotational Axis로 **0,0,1**으로 설정





Joint Creation – Translational Joint

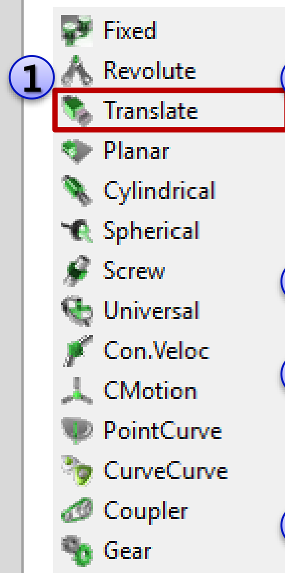
1. MBD Entities Toolbar에서 Joint – **Translate** 클릭

2. Detail 창에서 Connection Type 을 **Ground-Body** 로 변경

3. Action Body 로 ‘**Base**’ Geometry 선택

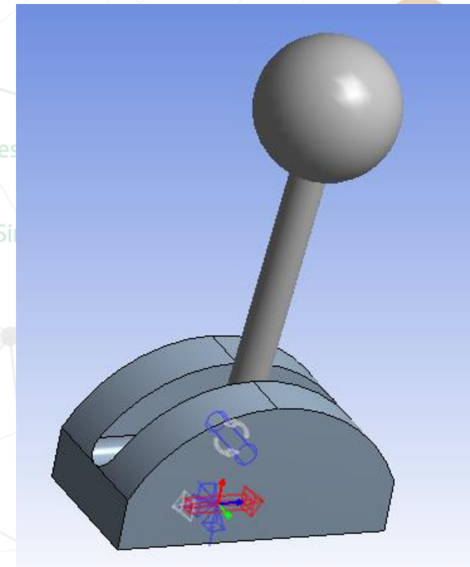
4. Origin 은 **0,-100,0** 으로 설정

5. Translational Axis로 **1,0,0** 으로 설정



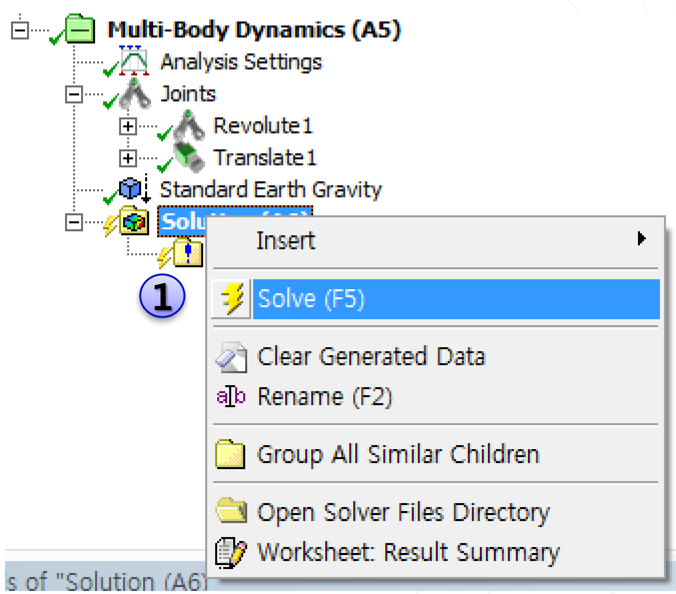
Details of "Translate1"

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

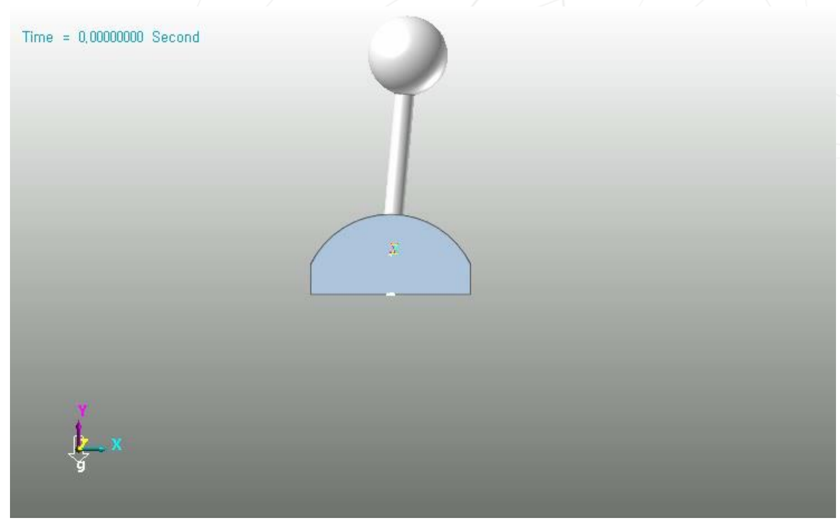


Simulation

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



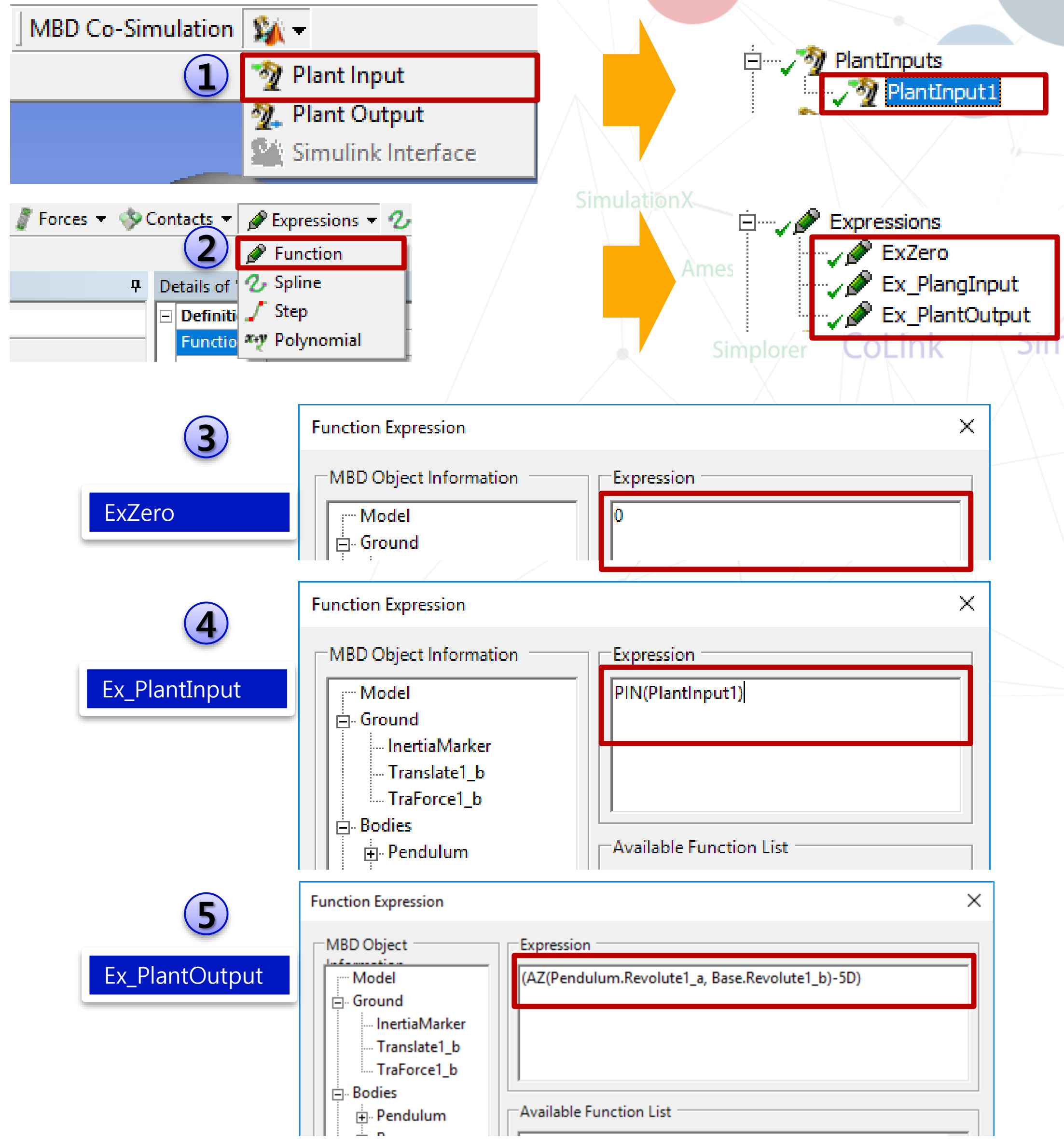
2



Play the animation

Plant Input, Expression 생성

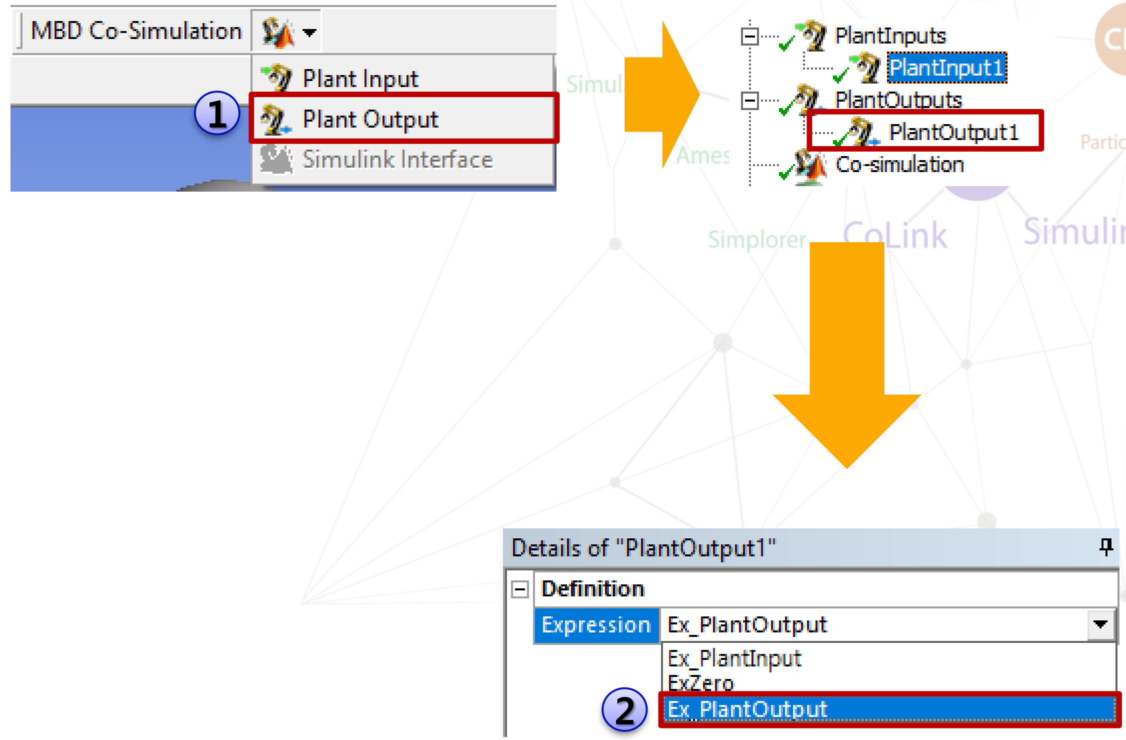
1. MBD Co-Simulation Toolbar에서 **Plant Input** 클릭
2. MBD Entities Toolbar에서 Expressions – **Function** 클릭
3. Expression 이름을 **ExZero**로 변경하고, Function Expression을 2개 더 만든 후, **Ex_PlantInput, Ex_PlantOutput으로 이름 변경**
4. ExZero 입력창에 **0** 입력
5. Ex_PlantInput 입력창에 **PIN(PlantInput1)** 입력
6. Ex_PlantOutput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



PlantOutput 생성

1. MBD Co-Simulation Toolbar에서 **Plant Output** 클릭

2. Detail 창 에서 Expression 으로 **Ex_PlantOutput** 선택



Force Creation – Translational Force

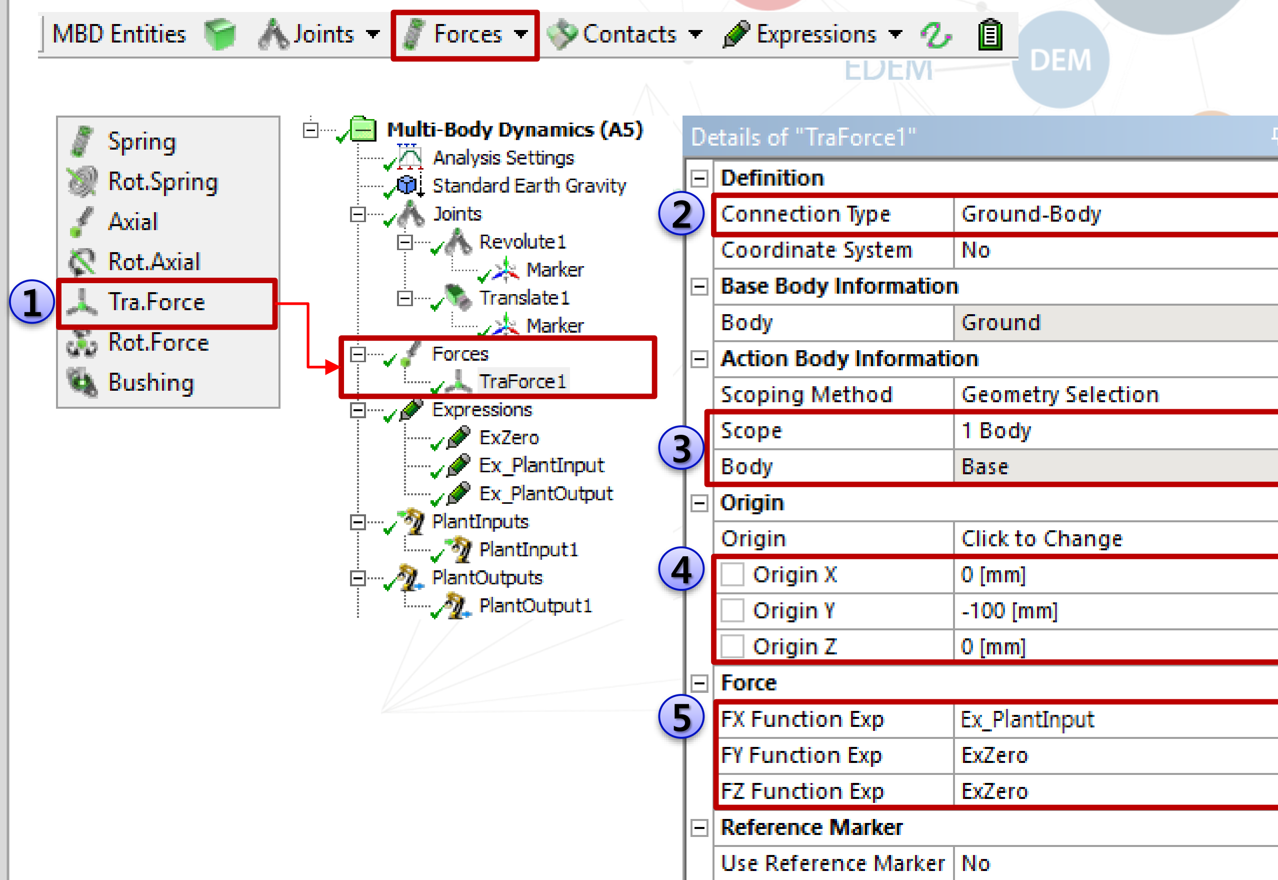
1. MBD Entities Toolbar 에서 Force – **Tra.Force** 클릭

2. Detail 창 에서 Connection Type 으로 **Ground-Body** 선택

3. Action Body 로 '**Base**' Geometry 선택

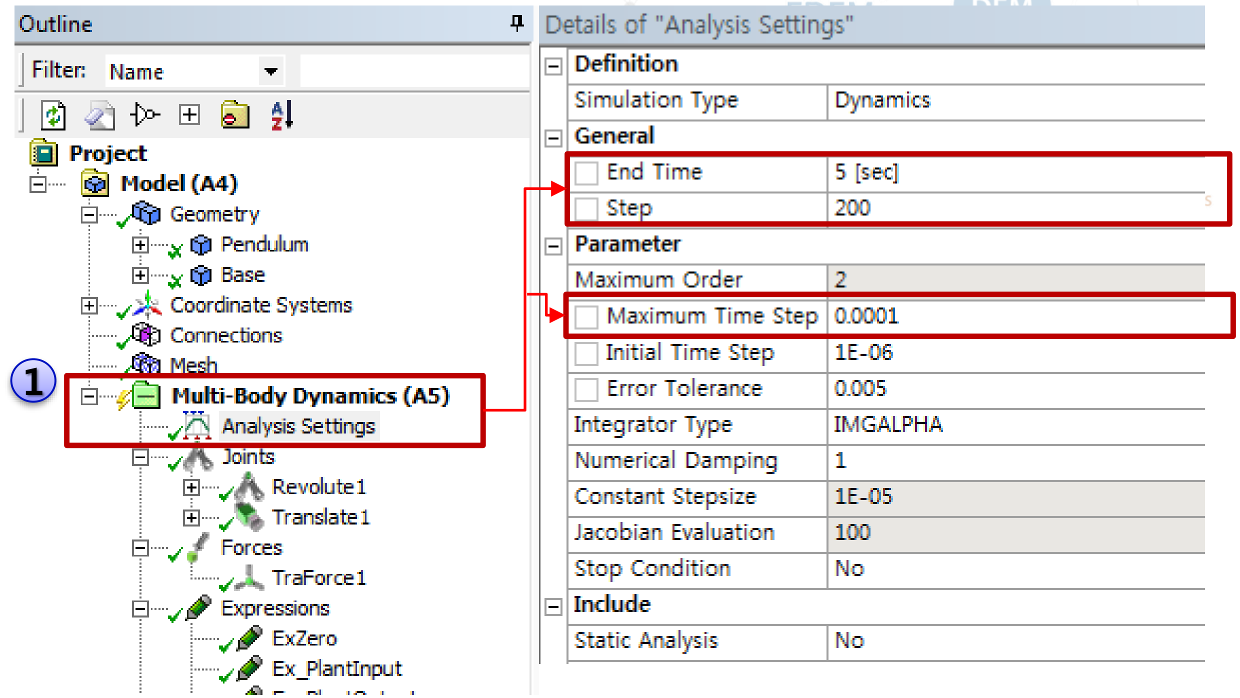
4. Origin 을 **0,-100,0** 으로 설정

5. Force Expression 에서 FX Function Exp 로 **Ex_PlantInput** 선택, 나머지는 **ExZero** 선택



Analysis Settings

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



Simulink Interface

1. MBD Co-Simulation Toolbar에서 **Simulink Interface** 클릭

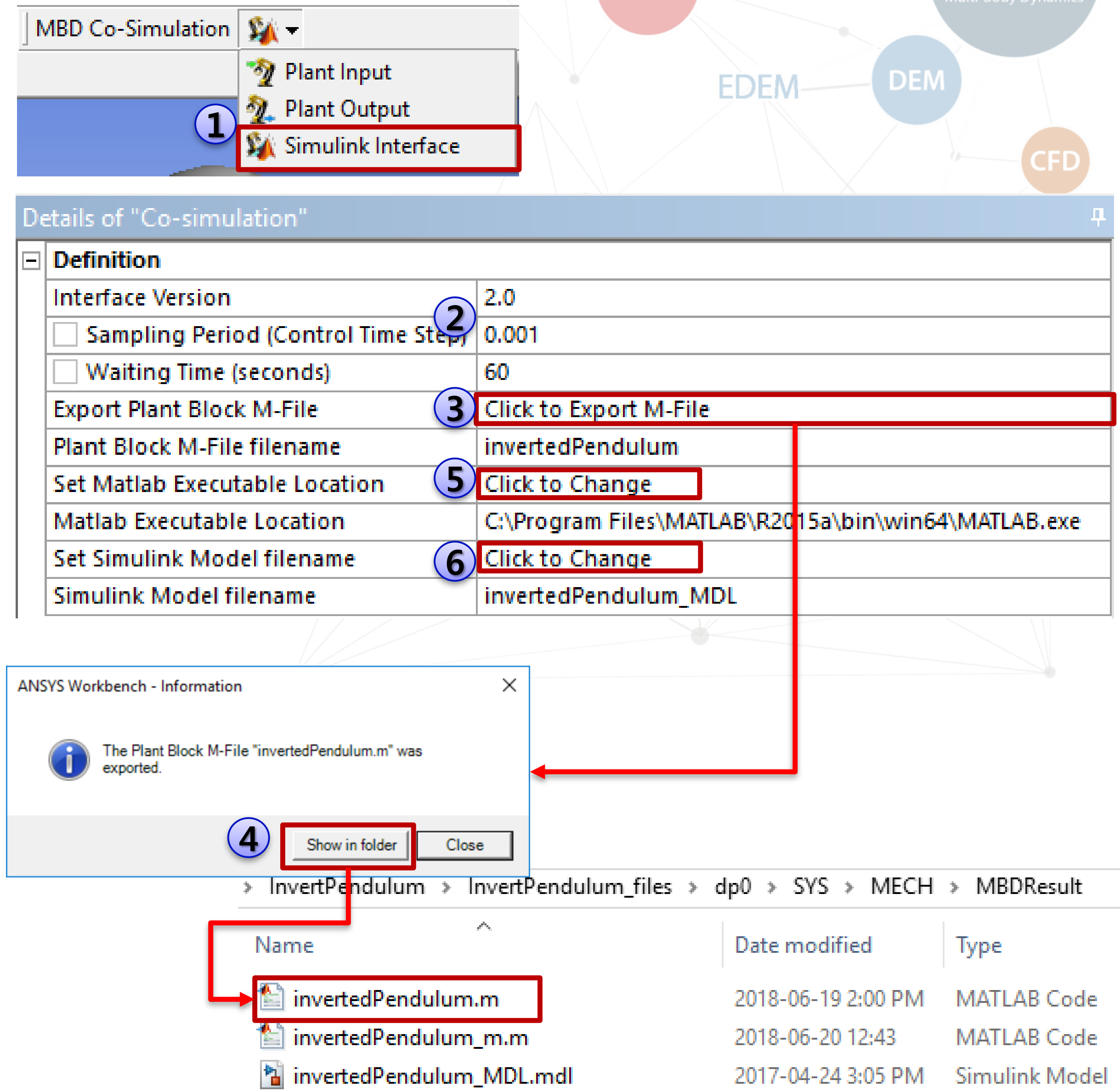
2. 아래와 같이 설정
   Interface Version = **2.0**
   Sampling Period = **0.001**
   (MATLAB 2016a 이상의 버전을 쓴다면 Interface Version = 3.0)

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 Location으로 Matlab.exe 위치 설정
   ($Matlabroot₩bin₩win**₩MATLAB.exe)

6. Simulink Model filename 으로 **invertedPendulum_MDL** 설정
   (MBD result 폴더에 있어야 함)



Matlab 모델링

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

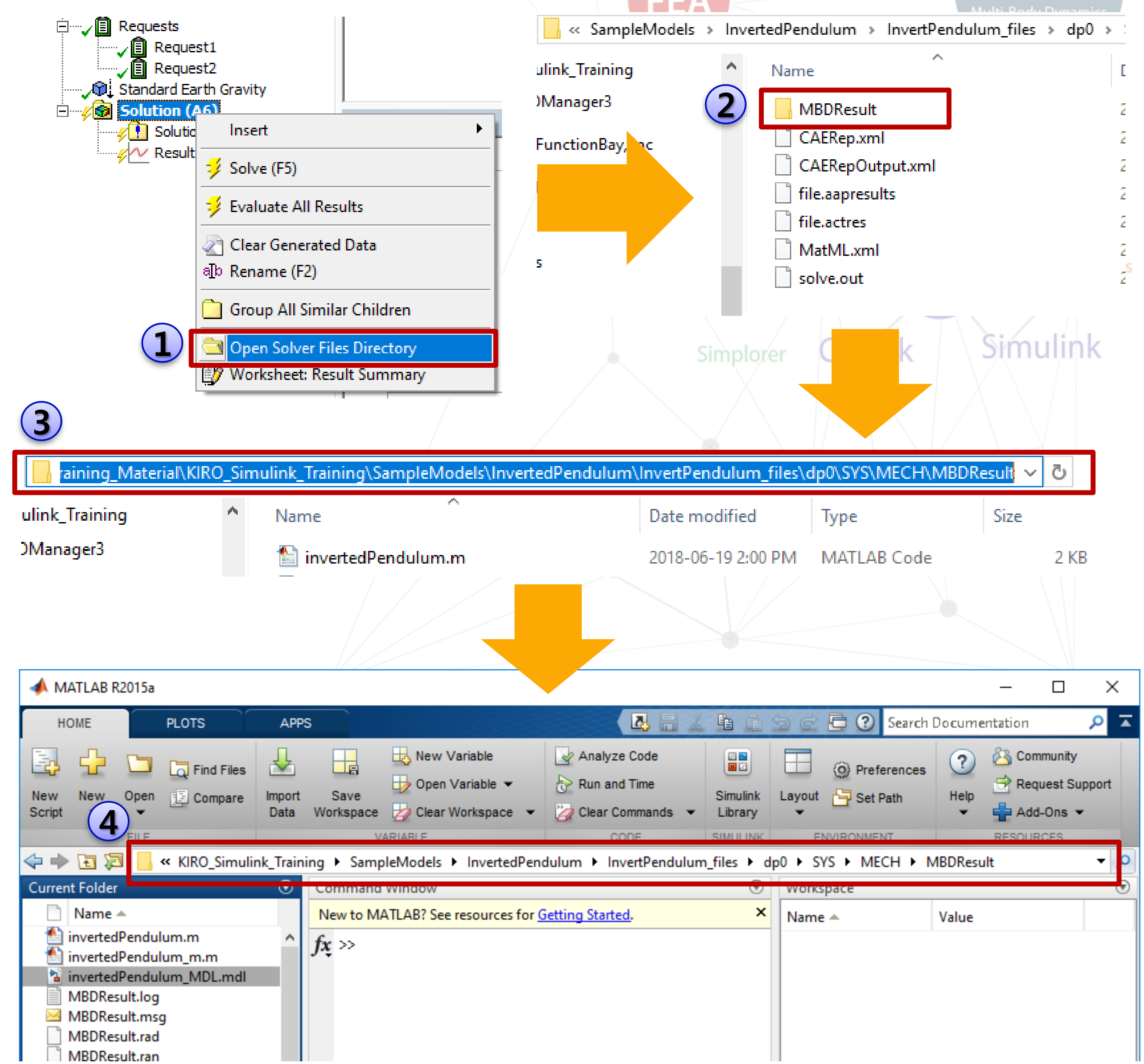
Set working directory

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

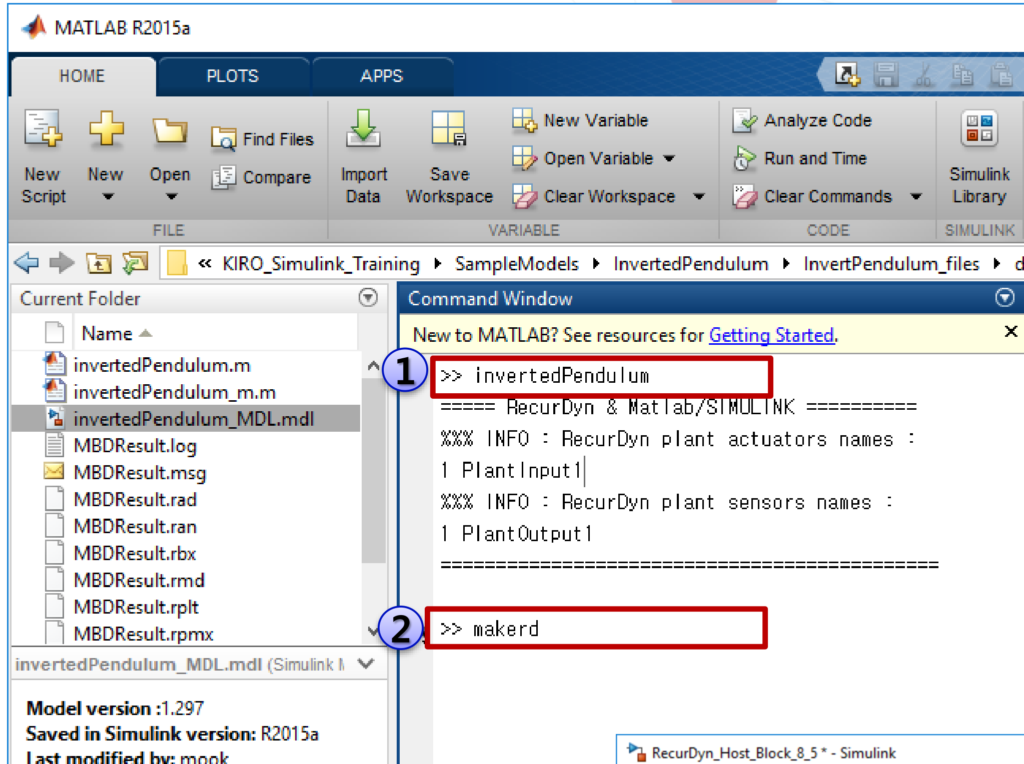


MBD for ANSYS Host Block 생성

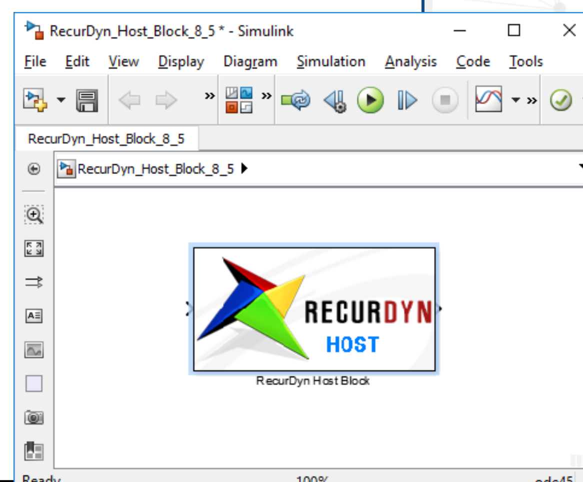
1. Command Window 에 **invertedPendulum** 을 입력하고 엔터

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.

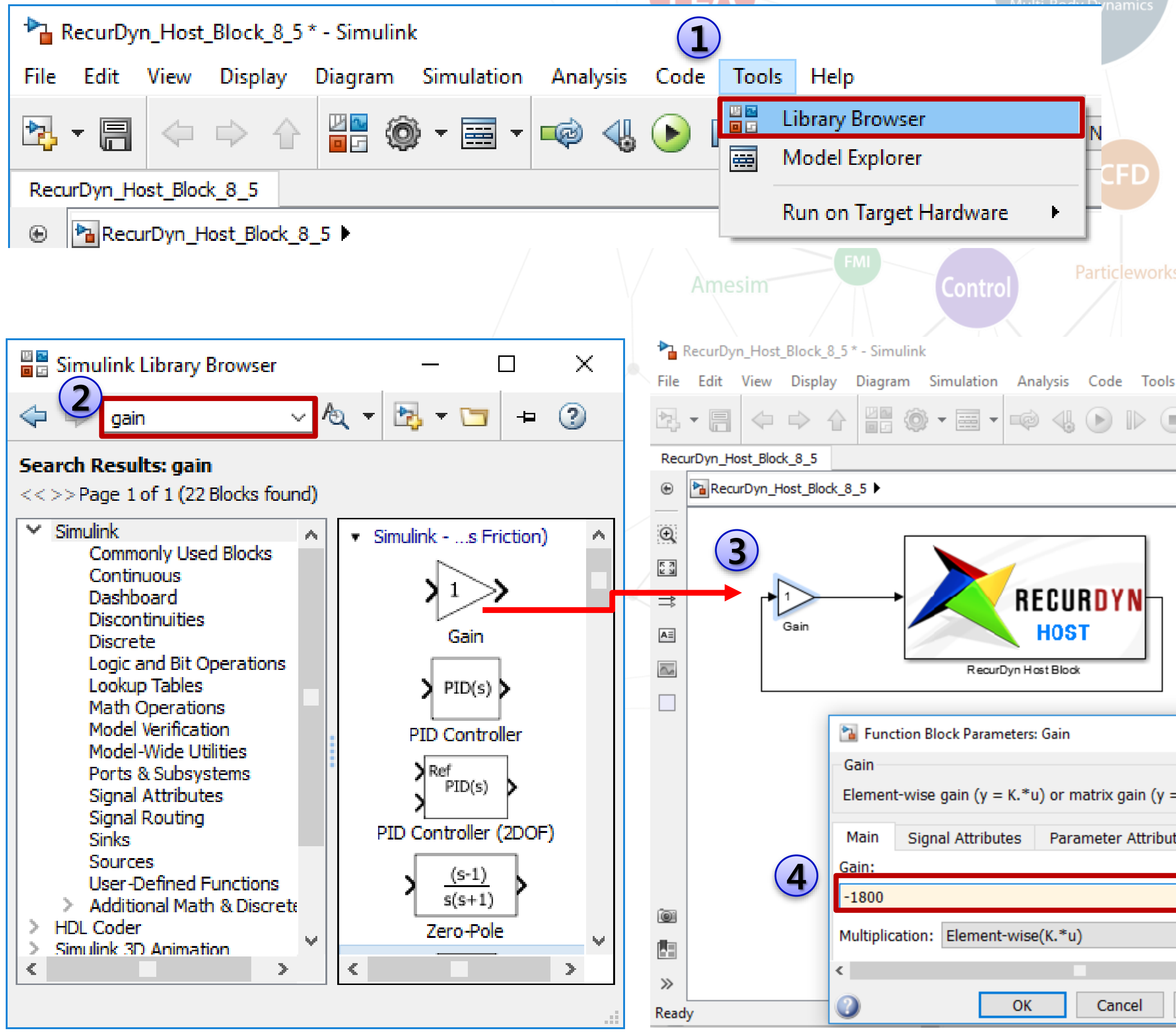
Simulink Block 생성

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

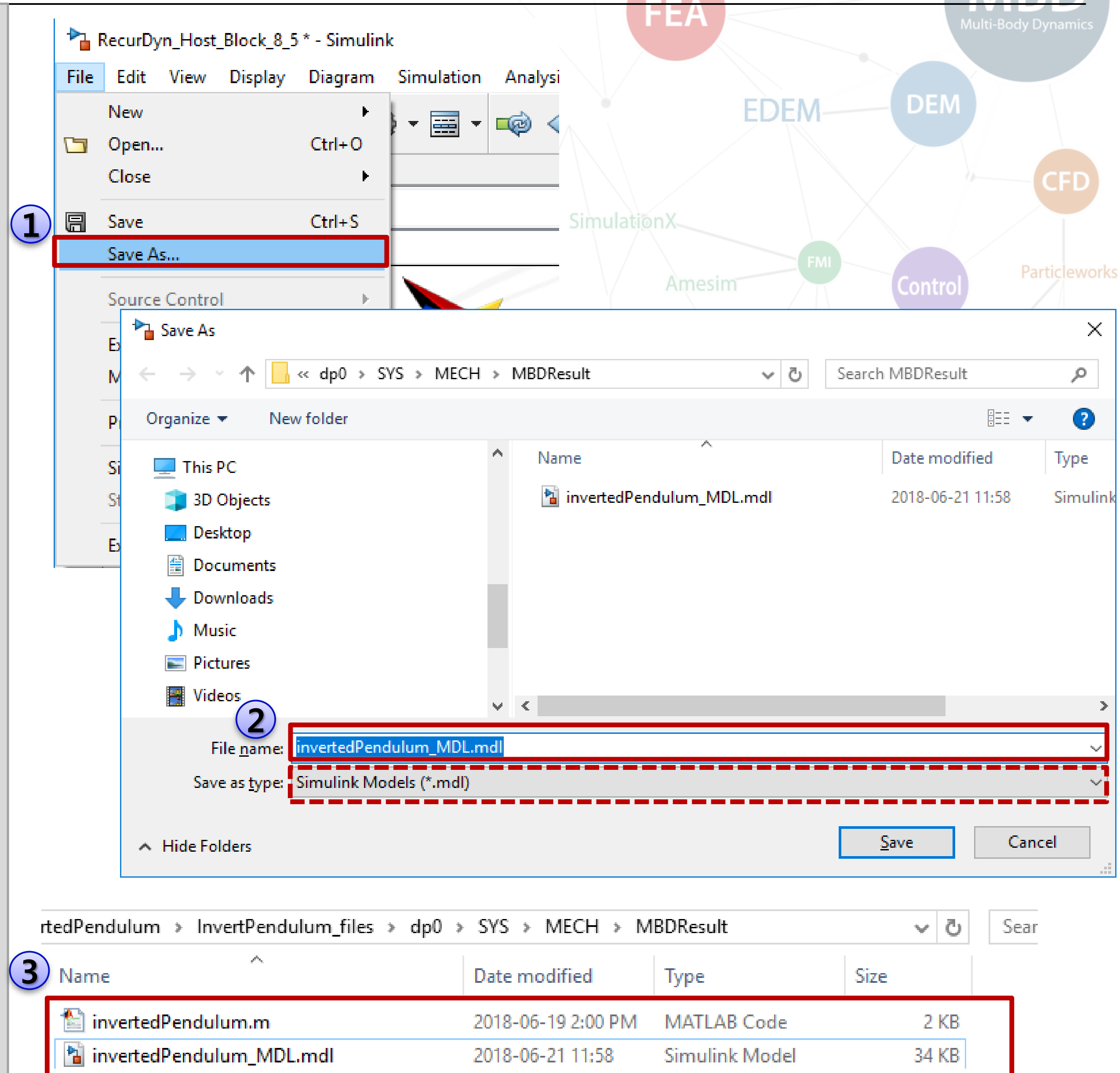


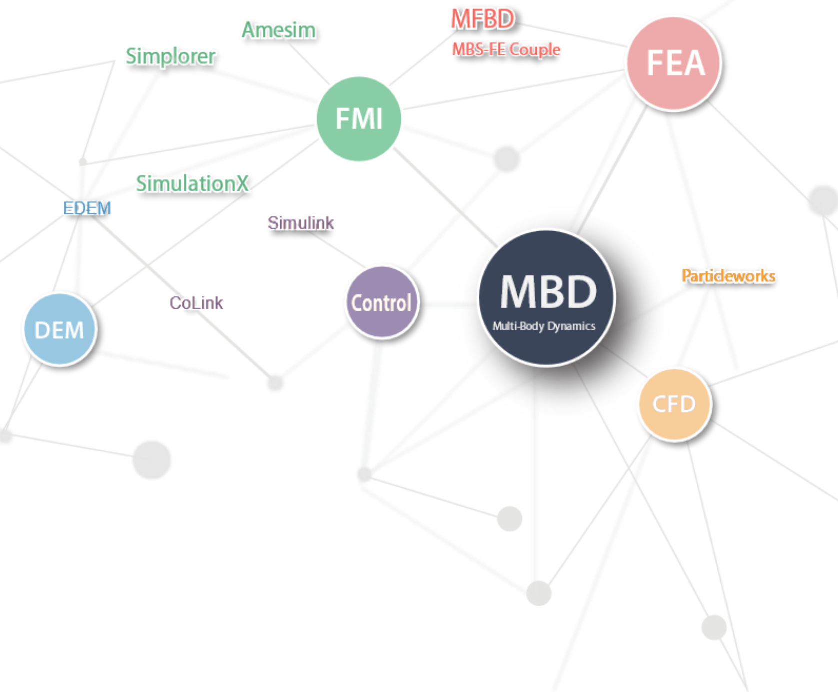
Simulink 모델 저장

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





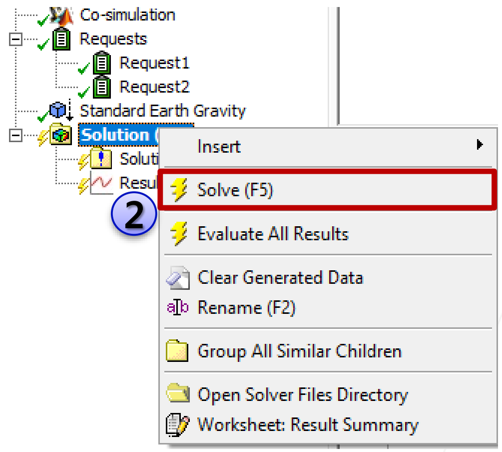
Simulation

1. Simulation
2. Post (Animation, Plot)

Simulation

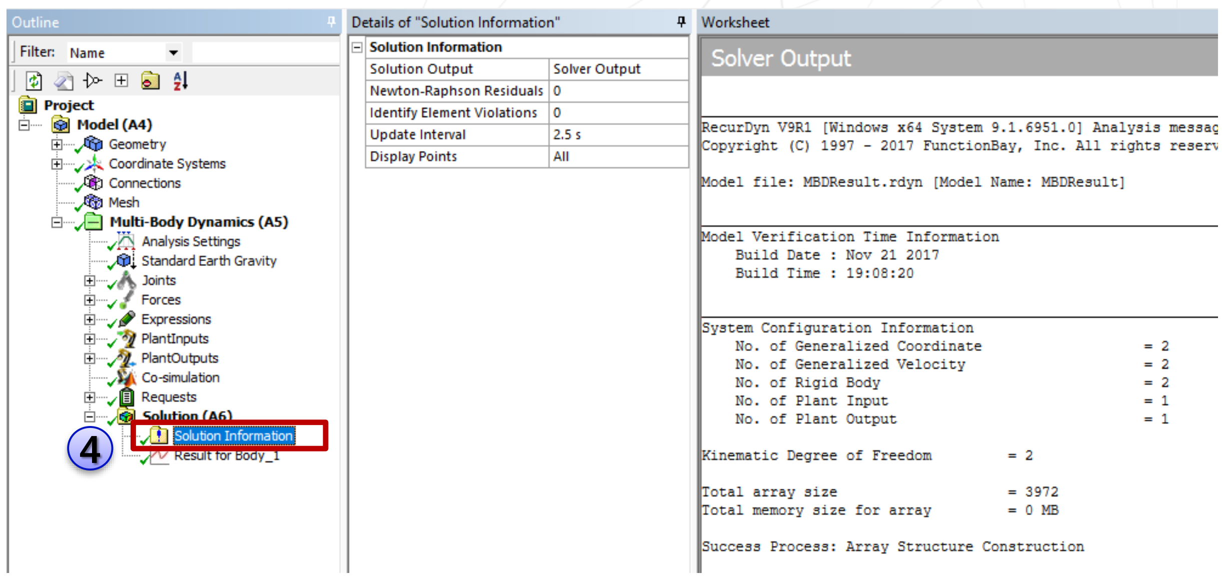
1. Simulation 하기 전에 **모델 저장**

2. Outline 에서 Solution 마우스 우 클릭 메뉴에서 **Solve** 클릭



3. Matlab 이 자동으로 실행되면서 Co-simulation

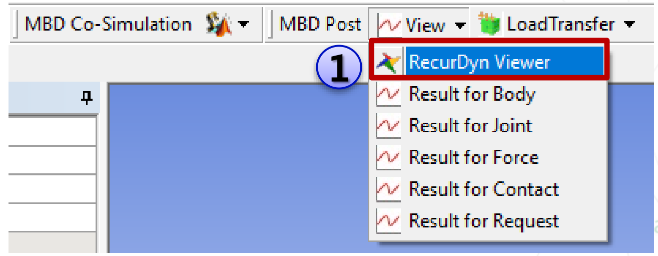
4. 해석이 완료 되면 녹색 체크 아이콘으로 바뀌고, Solution Information 을 클릭 하면 해석 결과에 대한 로그를 볼 수 있음

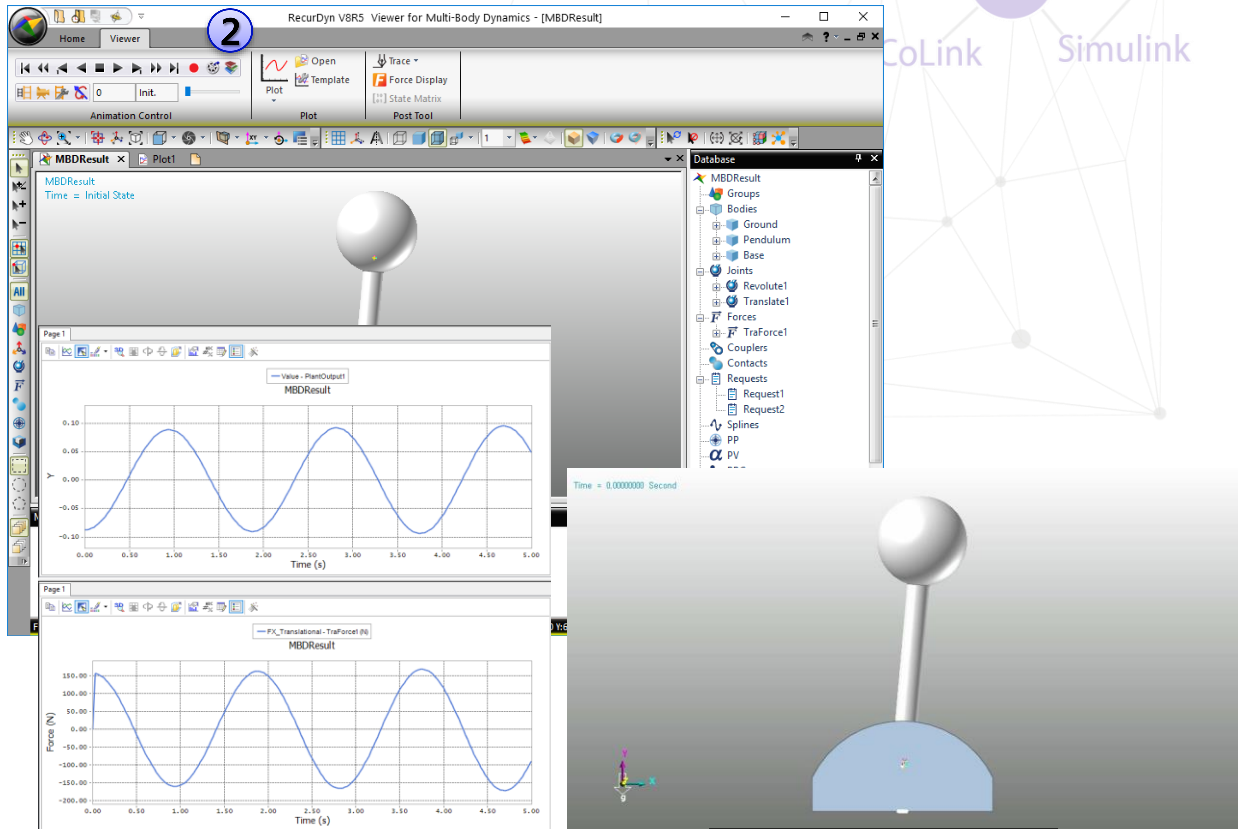


Post

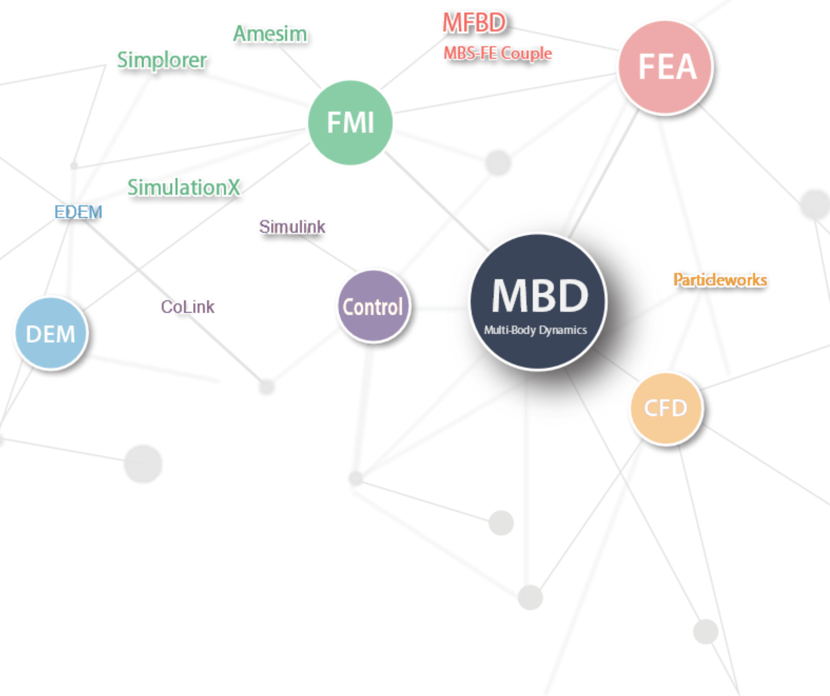
1. MBD Post Toolbar 에서 View – **RecurDyn Viewer** 클릭

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

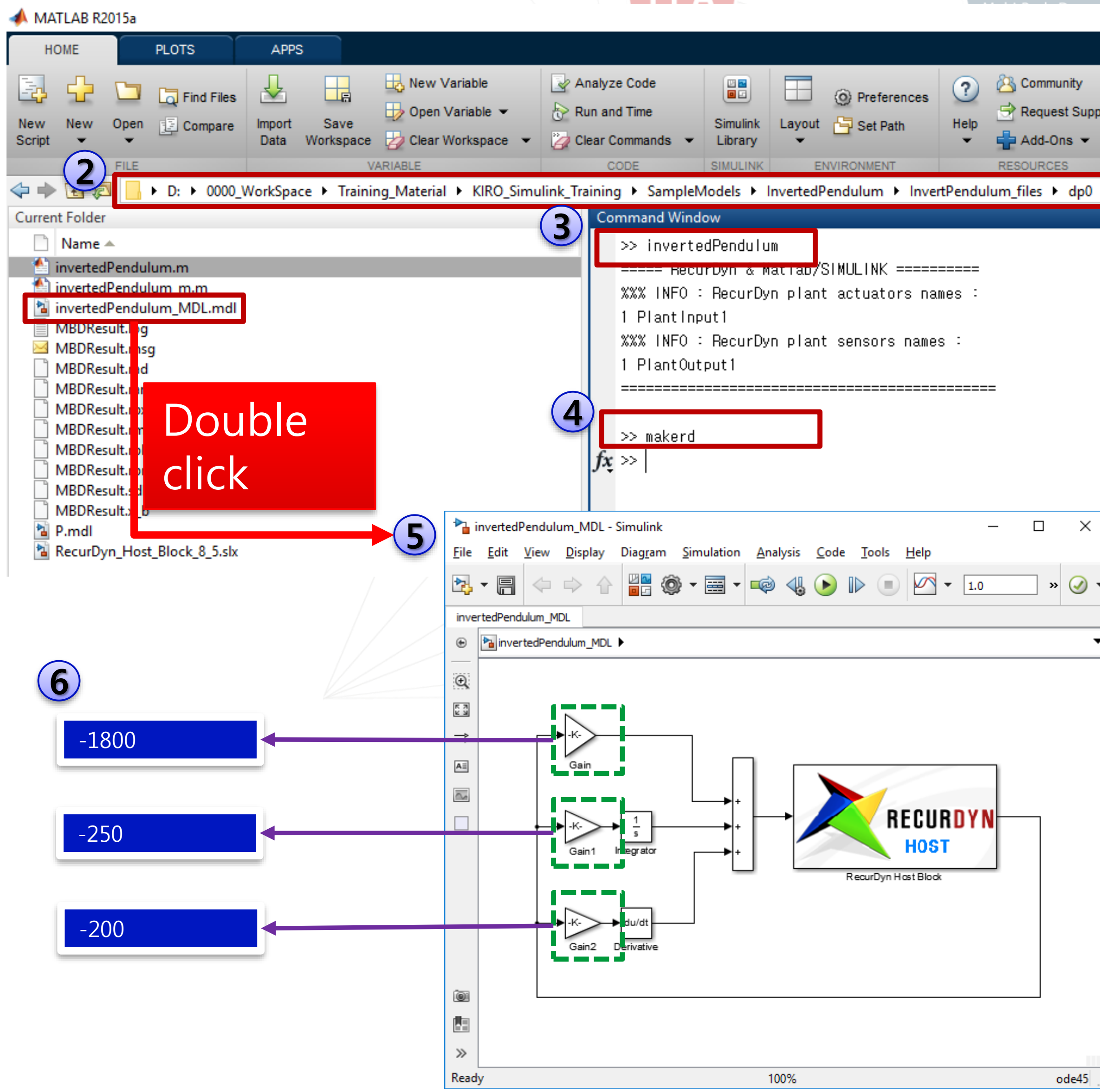
4. **Makerd** 입력하고 엔터

5. **invertedPendulum_M이.m이 을 더블 클릭**하여 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**

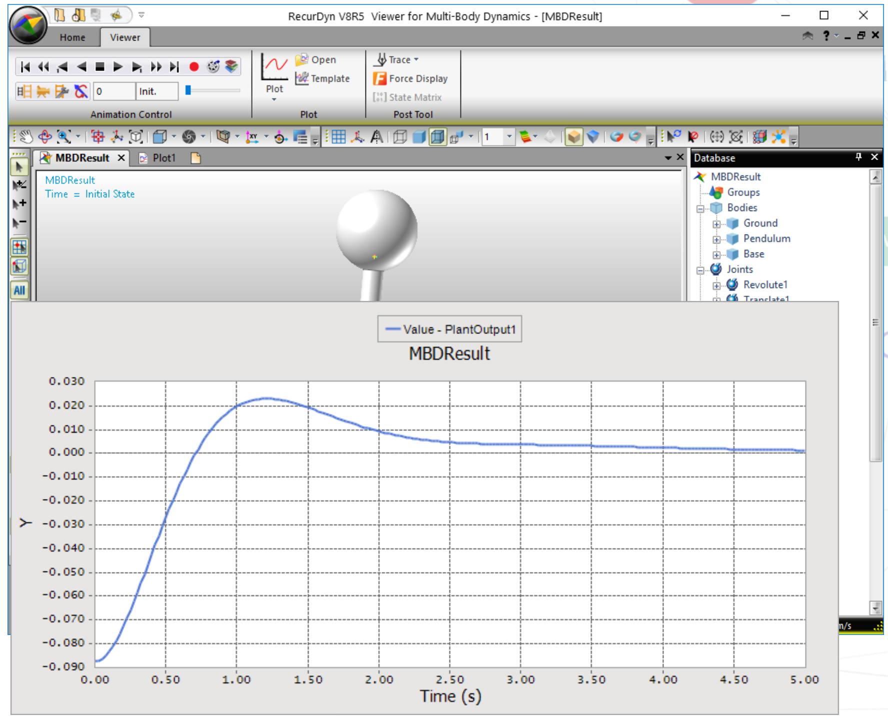
8. MBD for ANSYS에서 모델을 다시 시뮬레이션 (**Solve**)

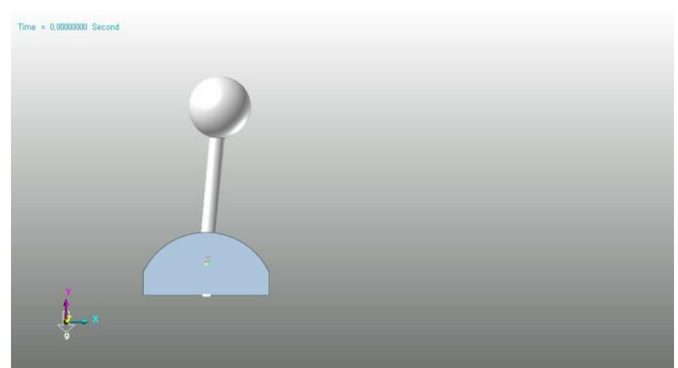


Post

1. MBD Post Toolbar 에서 View – **RecurDyn Viewer** 클릭

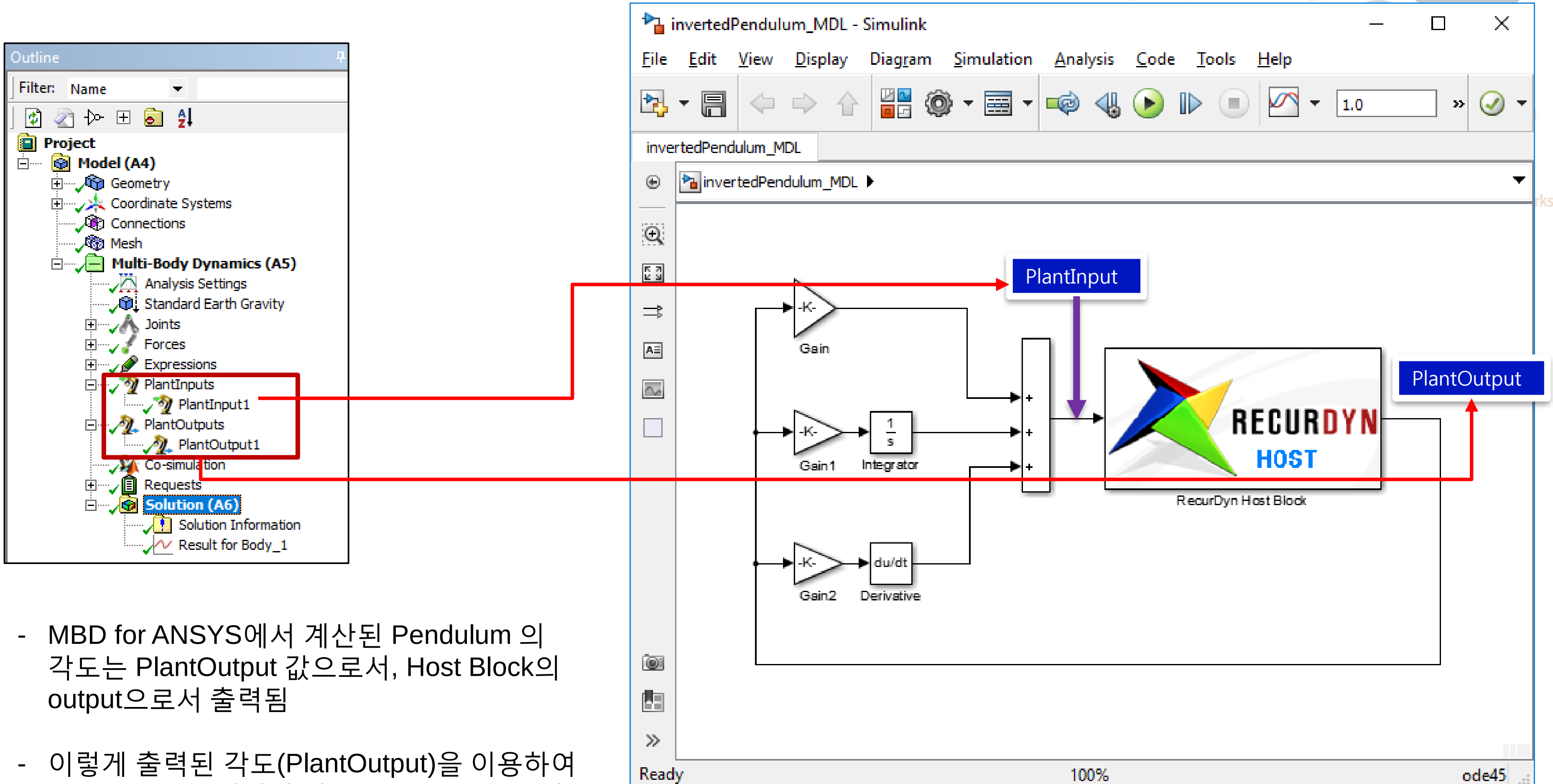
2. 변경된 결과 확인
    1. 애니메이션
    2. Plot





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