

예제로 쉽게 배우는 Ansys Motion

3. Robot Arm

Contents

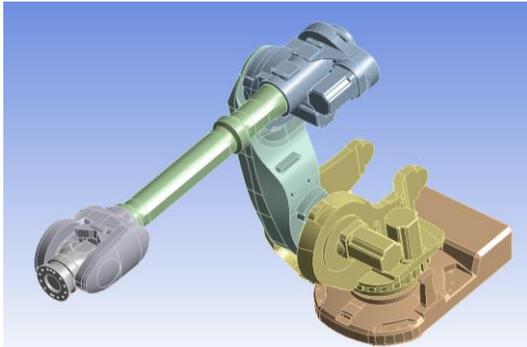
1. 개요
2. 전처리
3. 해석 & 후처리

1. 개요

1. 개요

목표

Workbench환경의 Ansys Motion으로
Robot Arm의 관절을 회전시키고,
End Effector의 이동 경로를 확인한다.

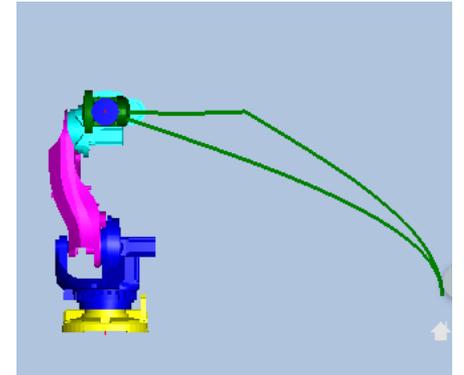


모델의 구성 요소

1. Body Type
 - Rigid Part : 7EA
2. Joint
 - Fixed Joint : 2EA
 - Revolute Joint : 5EA
3. Joint Load
 - Rotation : 5EA
4. Spline
 - Data set : 5EA

해석 결과

정기구학 분석을 통해
Robot Arm의 이동 경로 계산

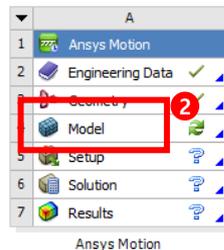
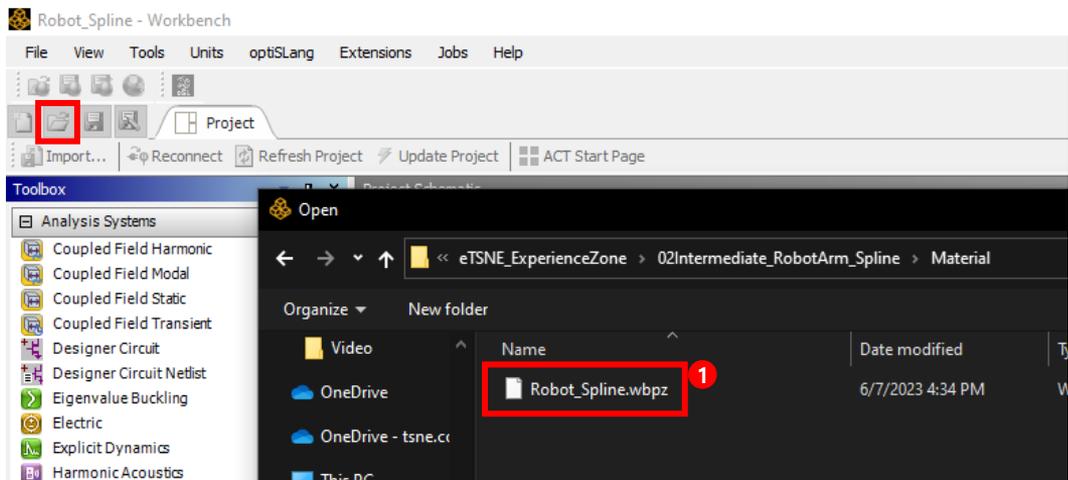


2. 전처리

2. 전처리

Body

1. Workbench 압축 파일 열기
- Robot_Spline.wbpz
2. Model 실행하기



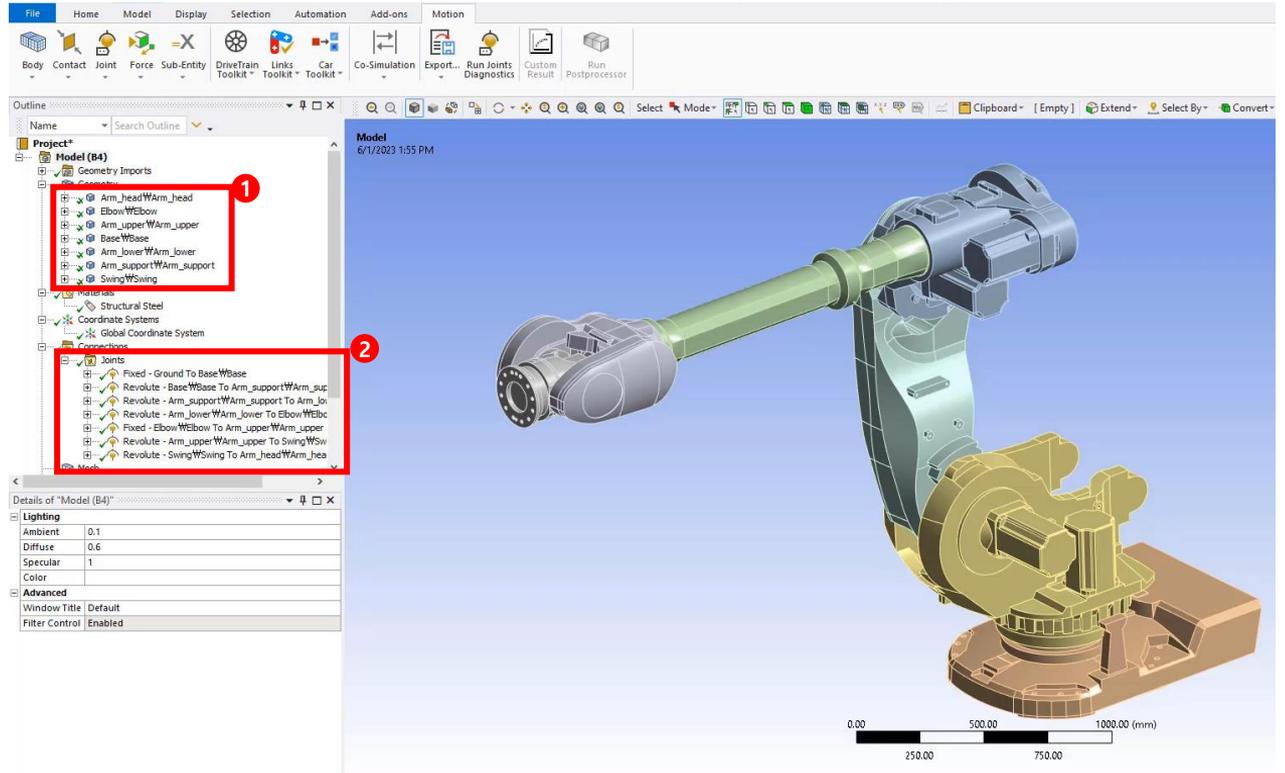
2. 전처리

Connections

1. Body 확인하기

2. 조인트 확인하기

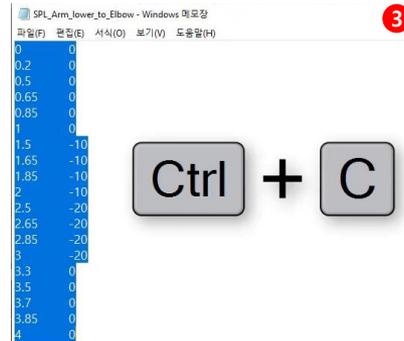
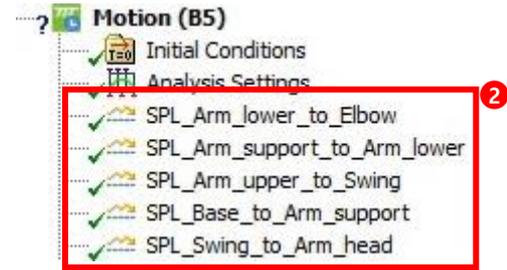
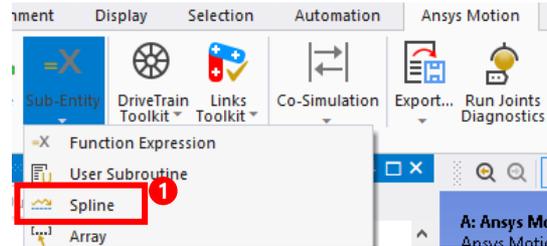
- Fixed Joint 2 EA
- Revolute Joint 5EA



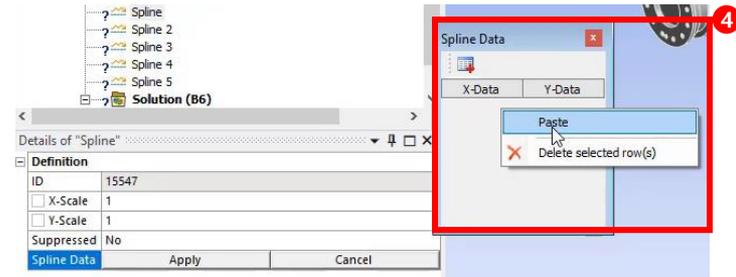
2. 전처리

Spline

1. Spline 생성하기 : 5 EA
2. 텍스트 파일에 맞게 Spline 이름 수정하기
3. 텍스트 파일 데이터 복사
4. Spline Data에 붙여넣기



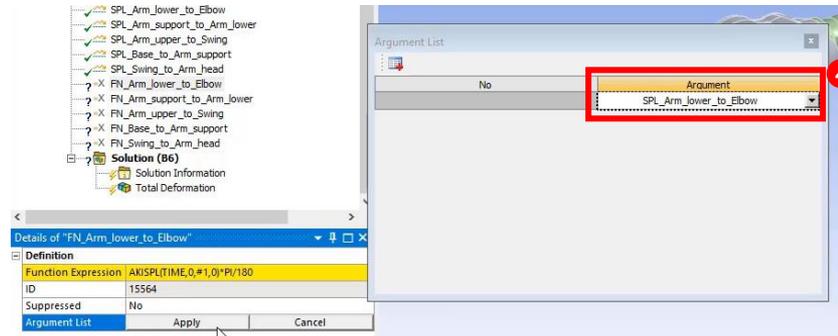
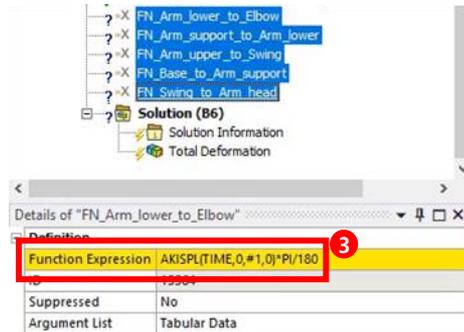
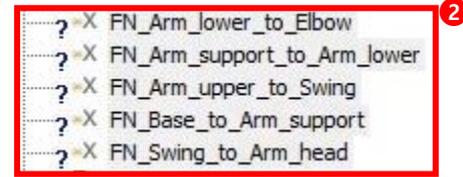
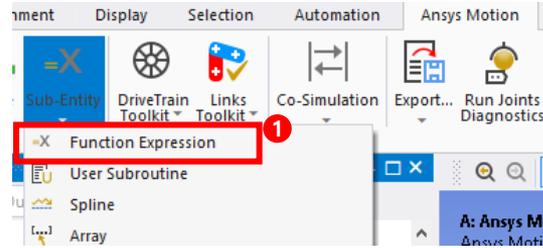
Ctrl + C



2. 전처리

Function Expression

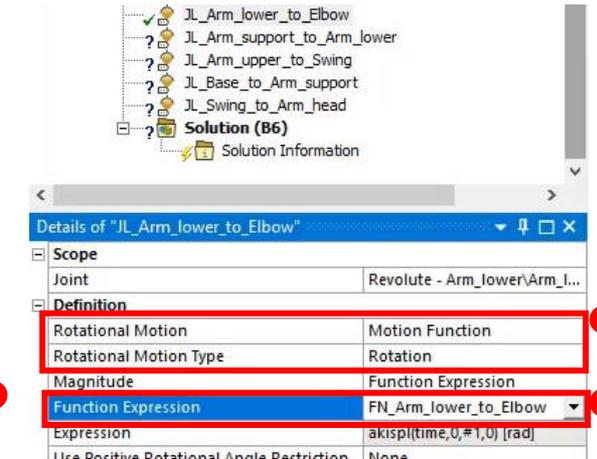
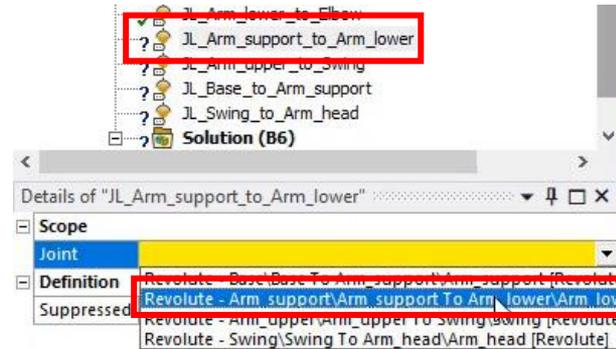
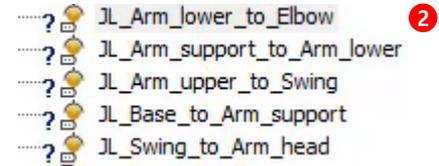
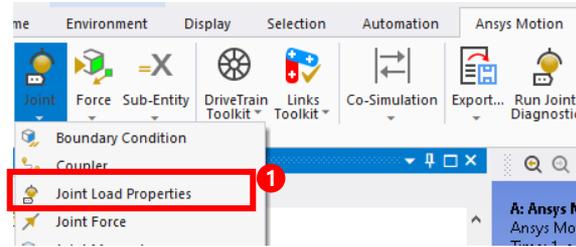
1. Function Expression 생성하기: 5 EA
2. Spline이름에 맞게 함수 명 변경하기
3. Function 5개 선택 후, 내용 입력
AKISPL(Time, 0, #1, 0) * Pi / 180
4. Function 이름에 맞는 Spline 선택
(5번 반복 작업)



2. 전처리

Joint Load Properties

1. Joint Load Properties 생성하기 : 5 EA
2. Function이름에 맞게 Joint Load 이름 수정
3. Joint Load 이름에 맞는 Revolute Joint 선택 (5번 반복 작업)
4. Joint Load 5개 선택 후, 조건 정의
 - Motion Function
 - Rotation
5. Joint Load 이름에 맞는 Function 선택 (5번 반복 작업)

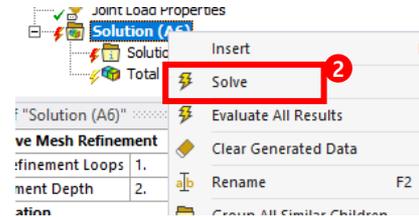
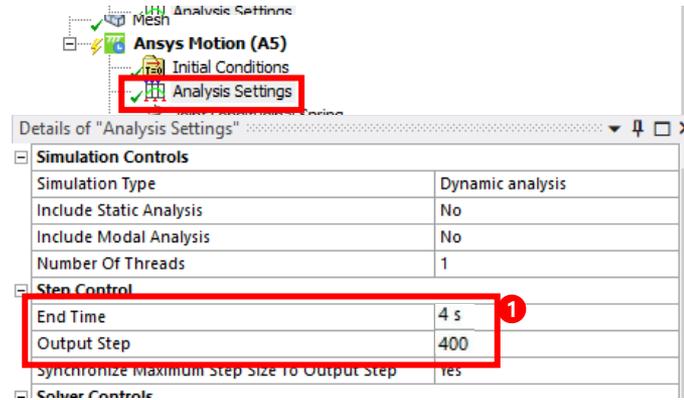


3. 해석 & 후처리

3. 해석 & 후처리

Simulation

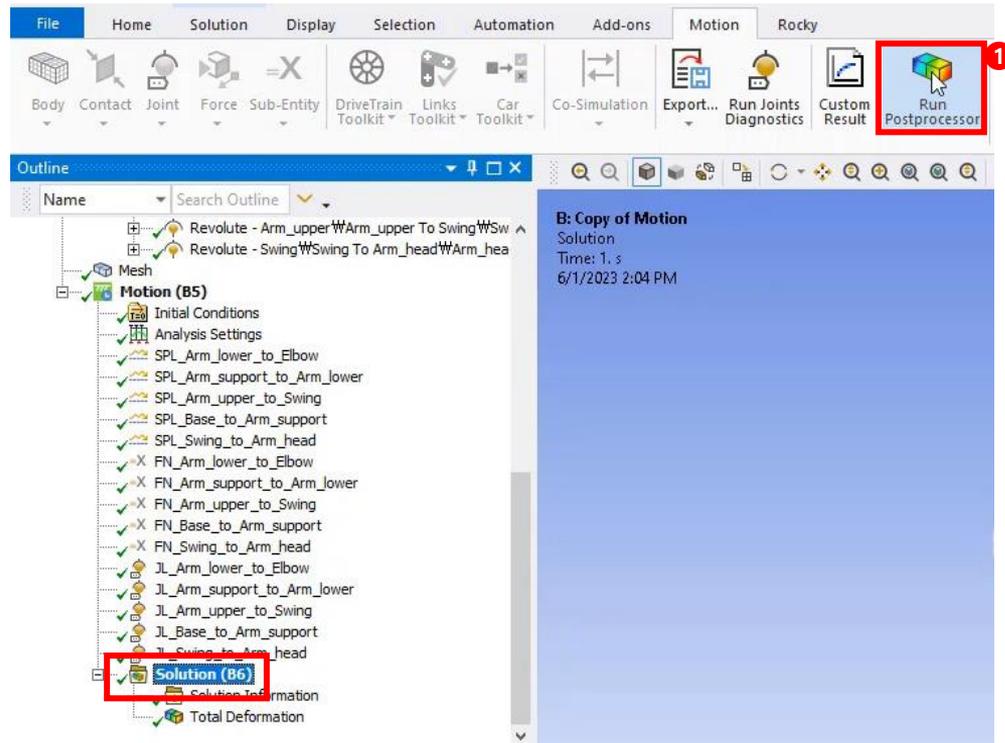
1. Analysis Setting
 - End Time : 4 sec
 - Output Step : 400
2. Solve



3. 해석 & 후처리

Post Process

1. Run Postprocessor 클릭



3. 해석 & 후처리

Post Process

1. Arm_head의 중심 마커 CM의 Trajectory를 Visible 활성화

