# Applying Multi-Flexbody Simulation to Non-Linear Joint Analysis

Presented by

**Paul D Fotsch**

&

SAE International

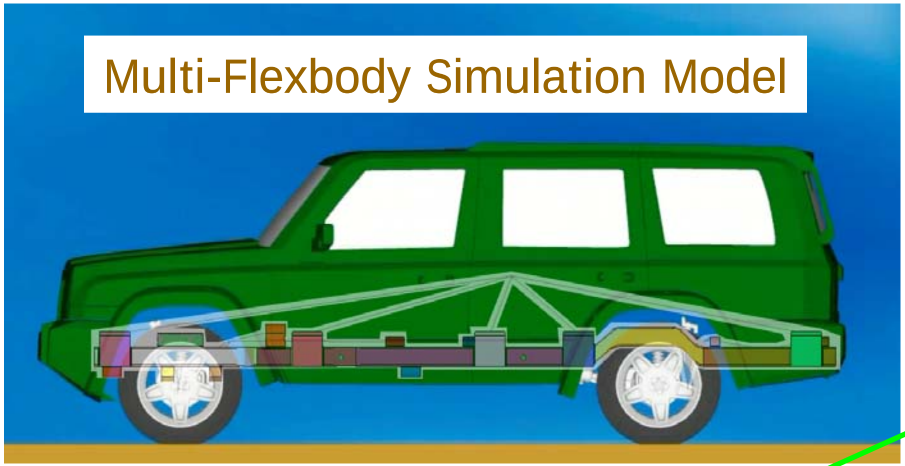
# Outline

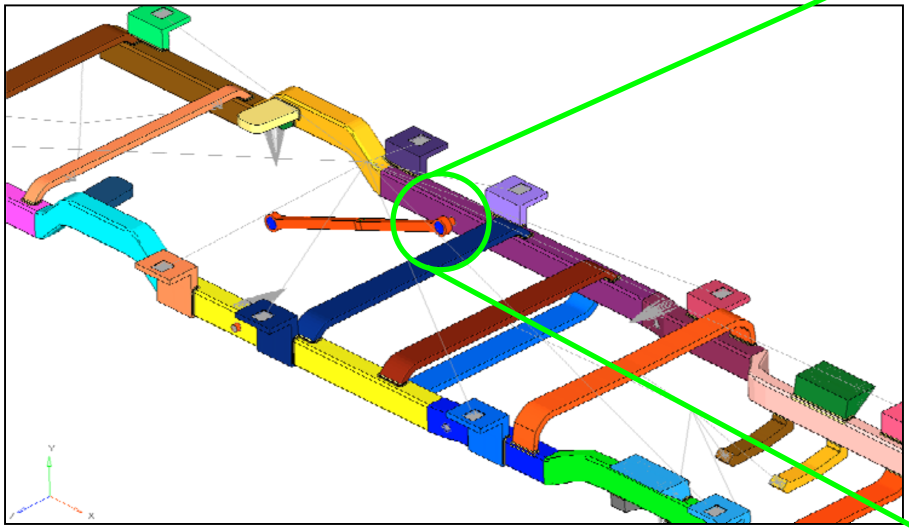
- Project overview
- Motivation
- Approach
- Linear Structural Model Preparation
- Cut-Surface Load Transformation
- Flexbody Simulation / Data Export
- Nonlinear Breakout Model
- Breakout Model Analysis
- Software Involved
- Academic Model
- Application Model
- Questions / Discussion

## Project Overview

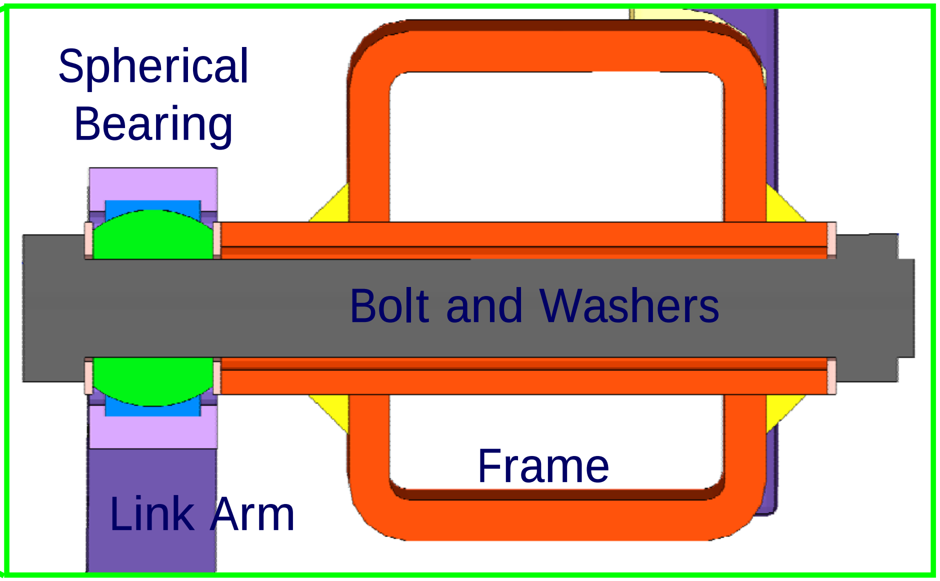
- Employ multi-flexbody simulation to compute quasi-static loads on a reduced Finite Element (FE) Model (FEM).
- Convert the reduced FEM loads to a meaningful load distribution on a detailed nonlinear analysis model.
- Render from the detailed nonlinear model all pertinent characteristics including load and stress distribution, contact pressure, friction, seal failure, wear, etc.

# System & Detail Models



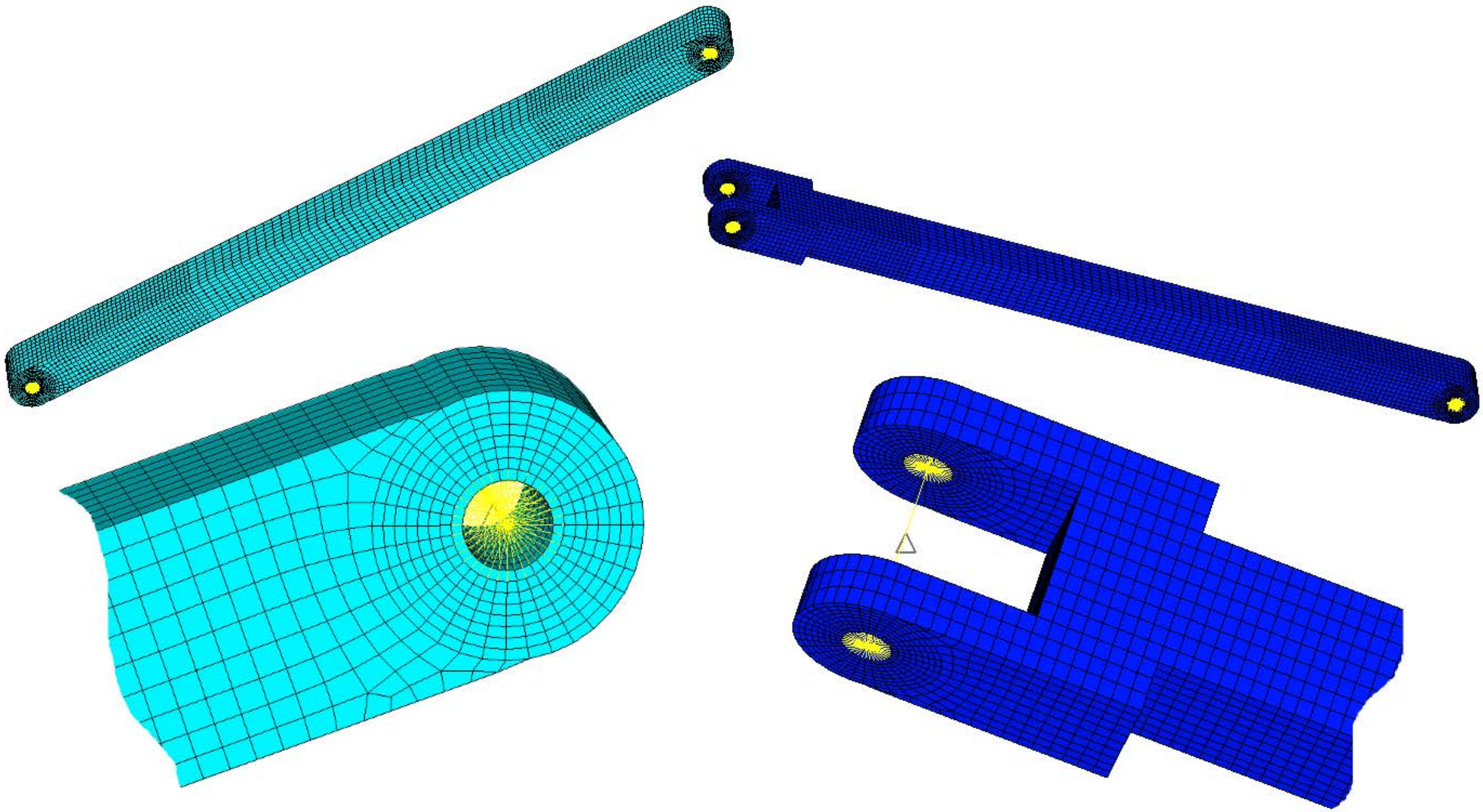
Detailed Nonlinear Model



# Motivation

- Multi-flexbody simulation has been shown to be effective for computing structural loads for mechanisms.
- Flexbody analysis requires a simplified representation at interfaces (joints) and loss of analysis effectiveness there.
- Hence, the goal is to employ flexbody simulation's ability to render loads away from an interface to perform a more effective analysis of a detailed interface model.

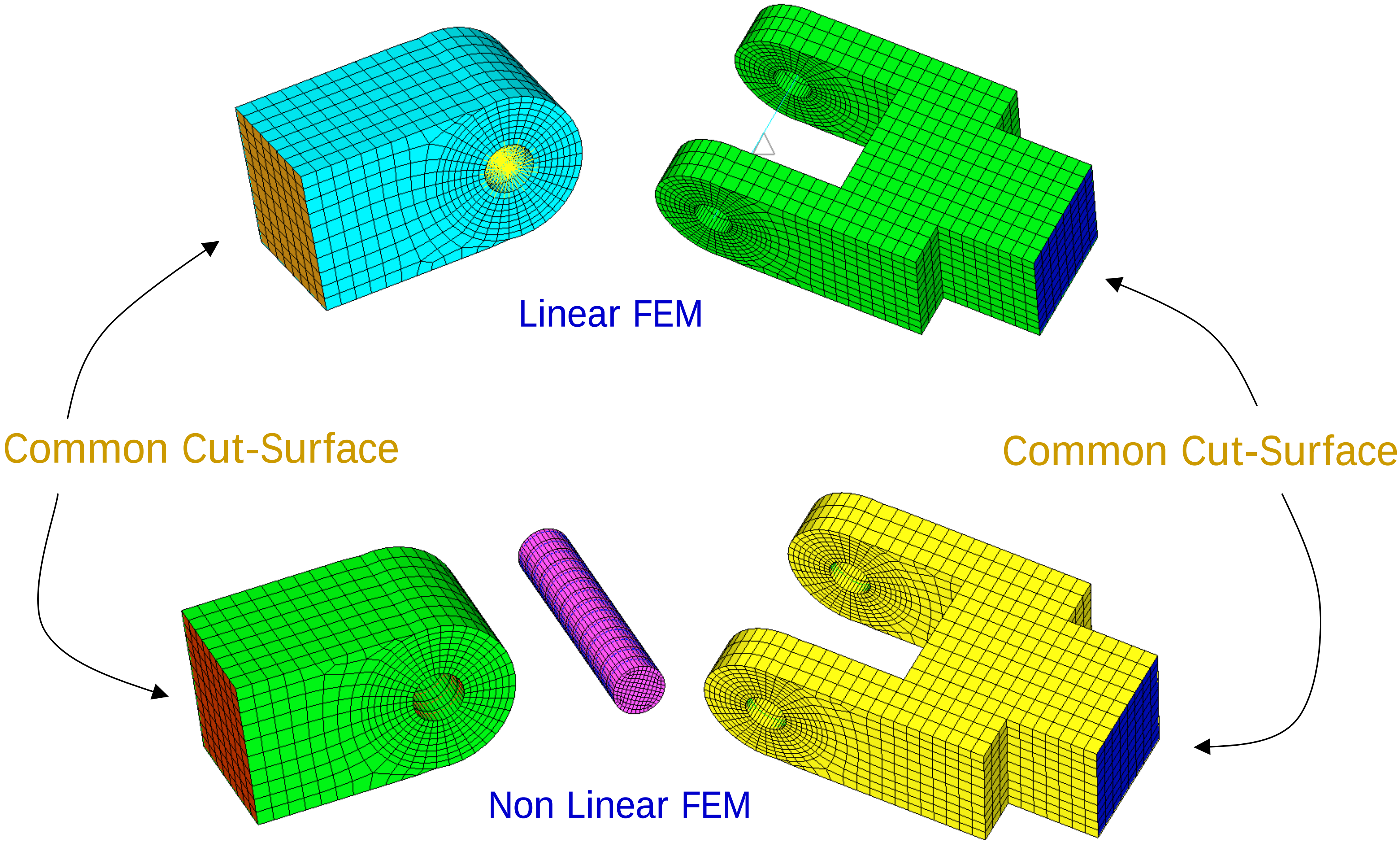
# Linear FEM for Flexbodies



## Approach

- Use a breakout model comprised of partial FE models of the interfacing structures for the detailed nonlinear analysis.
- Construct the FE models for both the flexbody simulation and nonlinear analysis to have a common boundary mesh.
- Compute load transformation matrices that convert flexbody loads to common boundary node loads during the flexbody creation process.

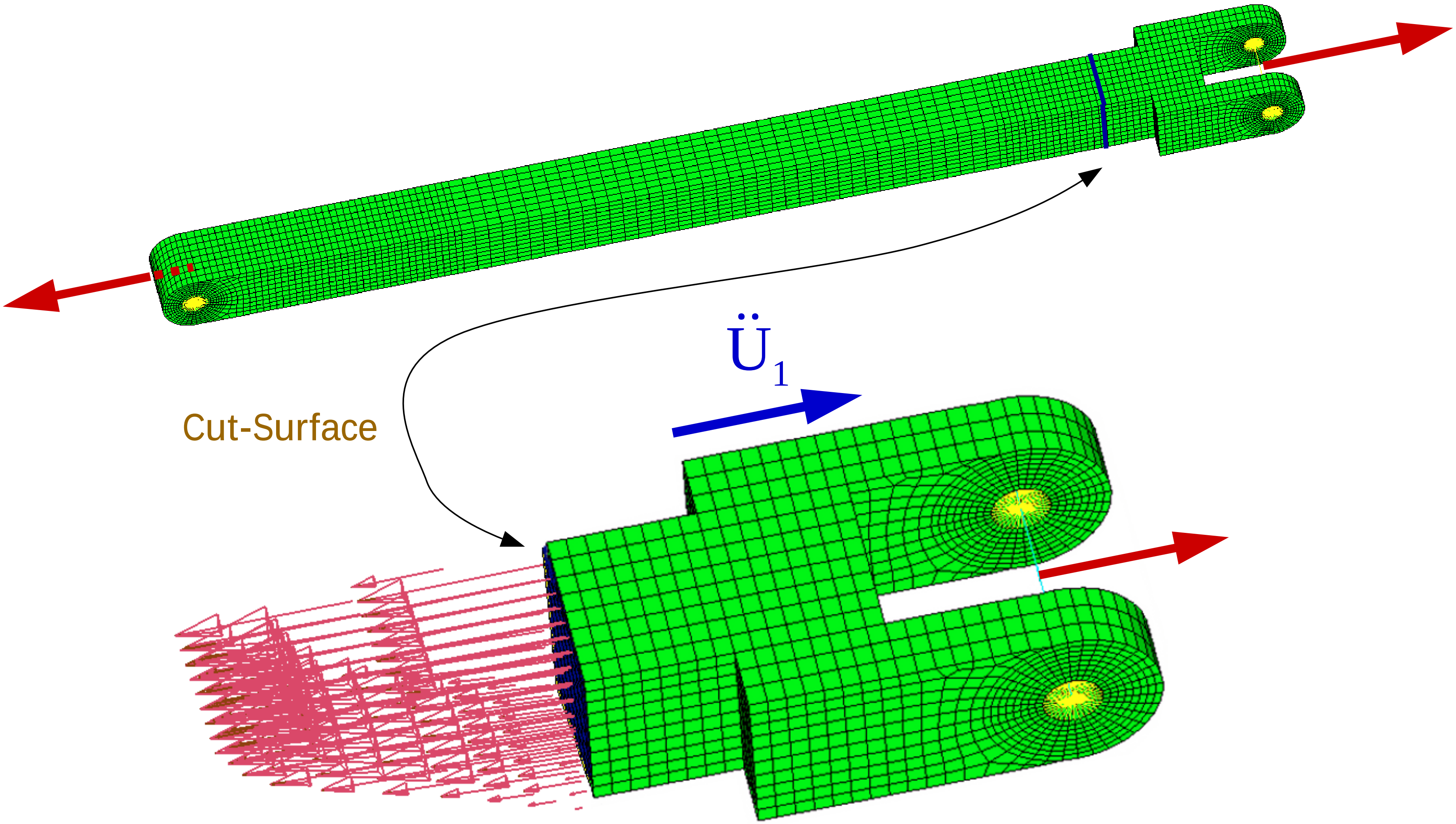
# Common Cut-Surfaces Between FEM



## Linear Structural Model Preparation

- Each structure involved in the interface is prepared for the flexbody simulation and breakout model analysis as follows:
  - Cut the FE model at a 'cut-surface' some distance from the interface to produce a 'cut-section'.
  - Simplify the interfaces using constraints, averaging elements, etc.
  - Mathematically reduce the model using Guyan reduction for flexbody use.
  - Perform an inertia relief analysis.
  - Construct a Force Transformation Matrix (FTM) to recover cut-surface forces from flexbody Degree of Freedom (DOF) forces.
  - Construct a Displacement Transformation Matrix (DTM) to recover cut-surface displacements from flexbody DOF forces.
- Note that the cut-surface is the common boundary mesh.

# Full Structure & Cut-Section Models



# Cut-Surface Load Transformation

- Apply unit loads to each simplified interface DOF as individual load cases.
- Compute the inertia relief response for each load case.
- Impose the displacement and acceleration results for each load case to the cut-section (partial) model according to:

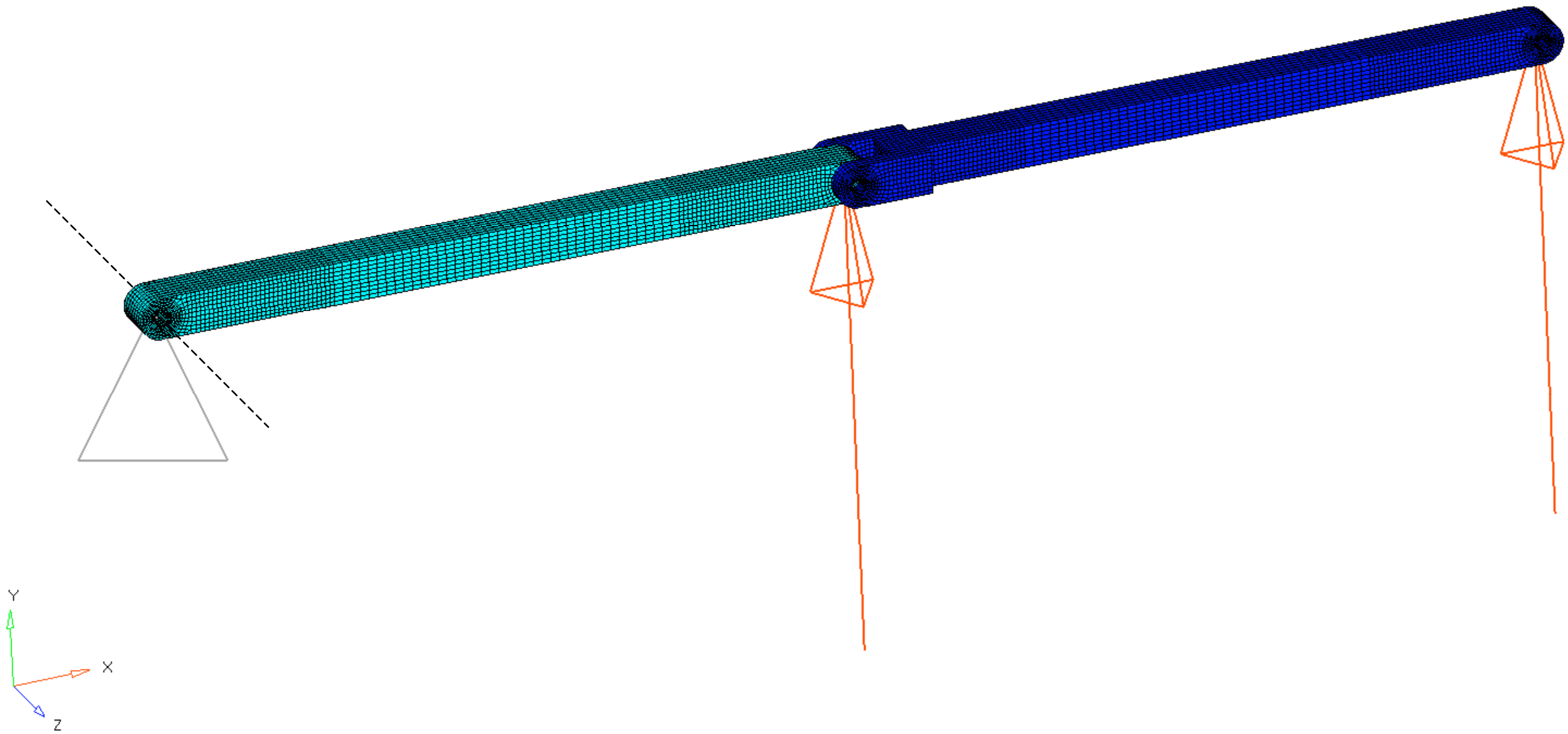
$$F_{sec} = M_{sec}\ddot{U}_{sec} + K_{sec}U_{sec}$$

- Extract cut-surface DOF entries from $F_{sec}$ and $U_{sec}$ and export them for use in loading the breakout model.

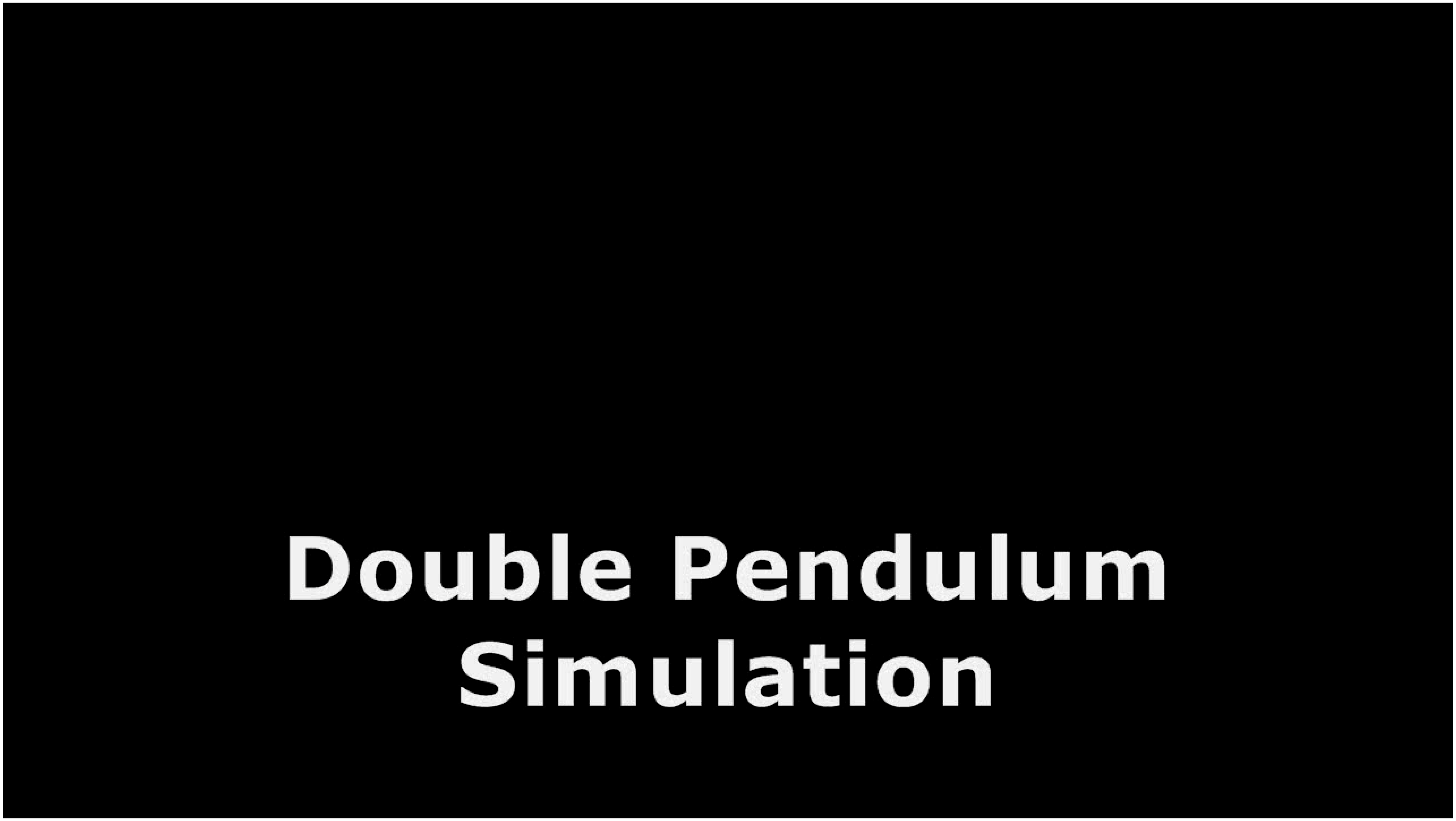
## Flexbody Simulation / Data Export

- Import the reduced FE models into the system model.
- Simulate desired operating conditions and expected events.
- Export the flexbody forces and body positions/orientations for select time points.

# Academic Model, Double Pendulum



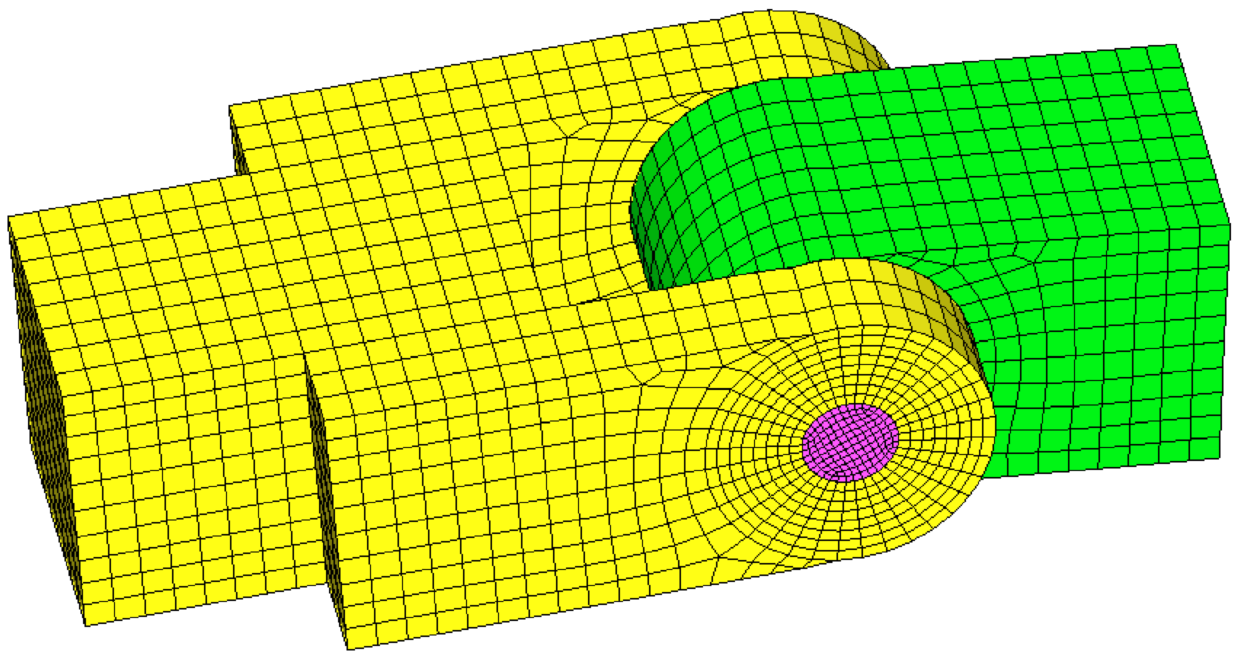
# Double Pendulum Simulation



## Nonlinear Breakout Model

- The breakout model replaces the simplified representation of the interface with one of great detail and refinement.
- Detail will generally include contact surfaces with friction and may include seals and other features.
- Additional parts may come into play as well.
- In short, all features that represent the real hardware may be modeled.

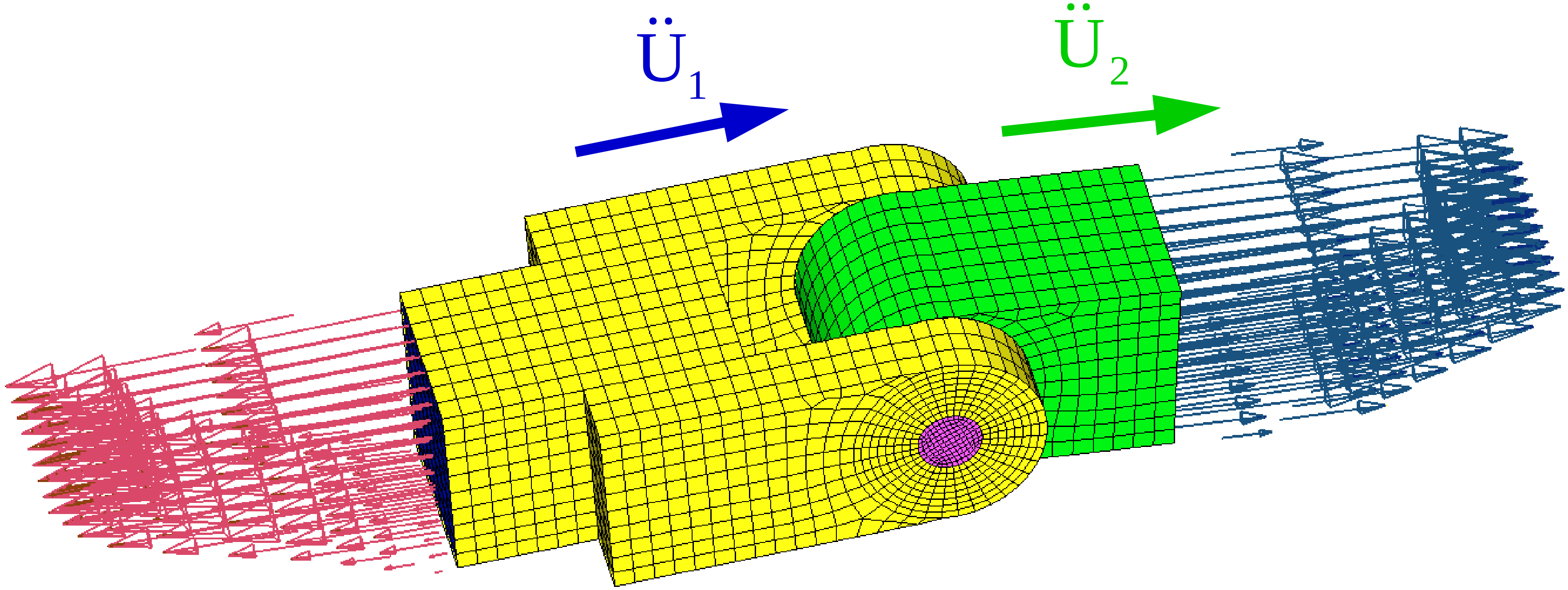
# Nonlinear Breakout FEM



# Breakout Model Analysis

- Loads are applied at the cut-surfaces based on the flexbody simulation loads and FTM produced in the FEM preparation step.
- Relative deformations are imposed at the cut-surfaces based on the flexbody simulation loads and the DTM produced in the FEM preparation step.
- Linear and rotational accelerations, based on the flexbody loads, are imposed on cut-sections and additional parts.
- Sufficient DOF are grounded to remove singularities.
- A ‘pseudo’ inertia relief analysis is performed.

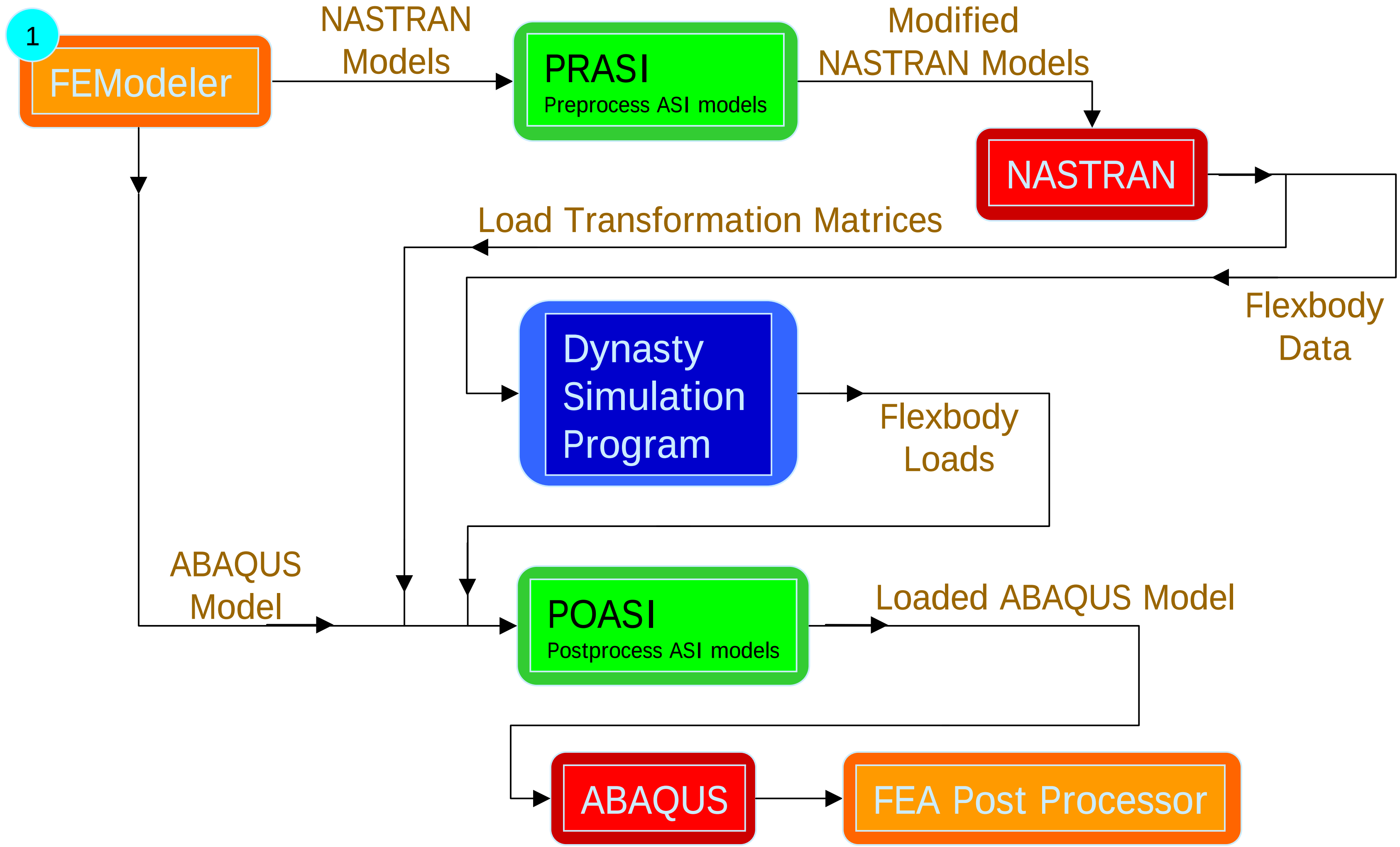
# Loaded Breakout Model



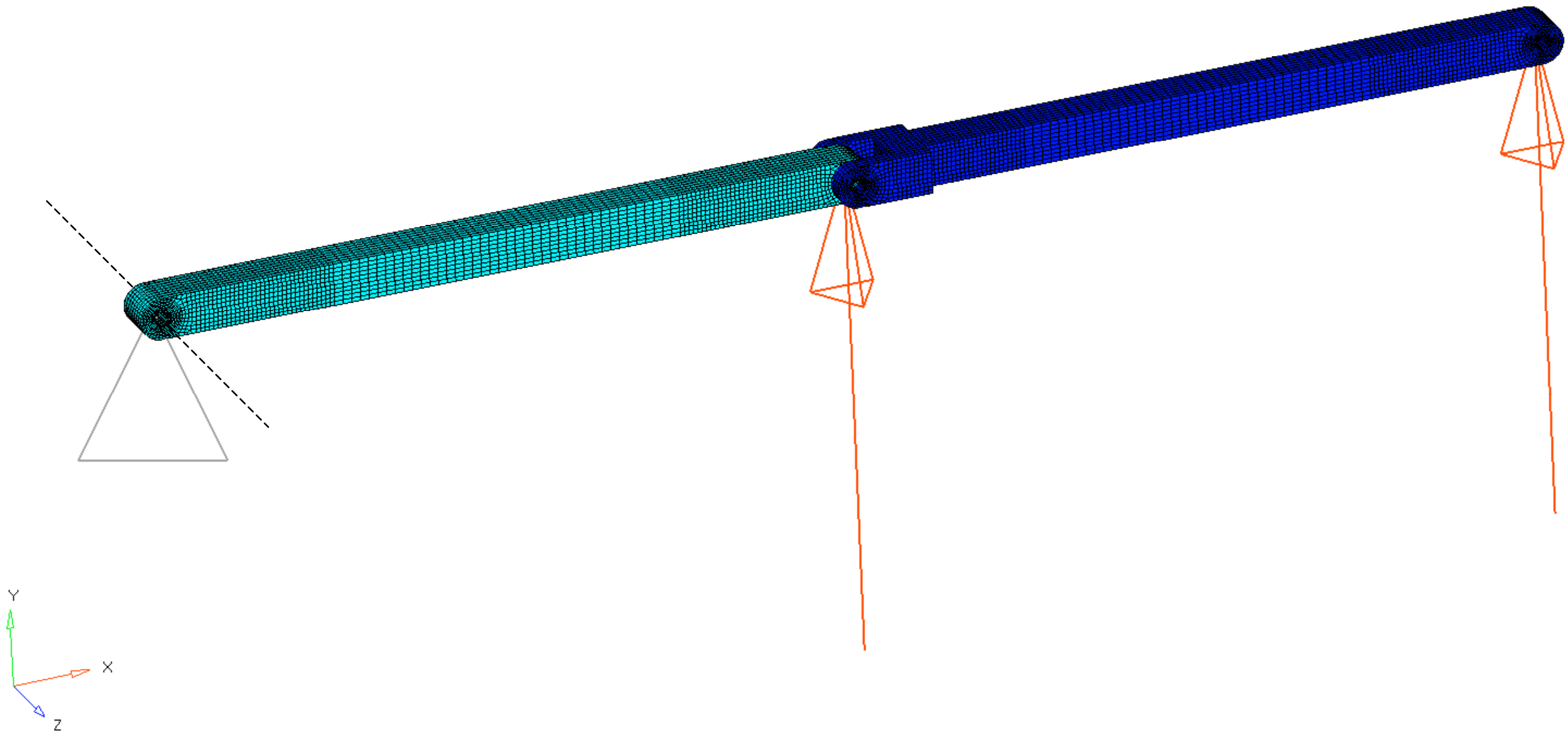
## Software Involved

- An in-house program, PRASI, processes cut-section NASTRAN models to produce a single structure's NASTRAN model.
- NASTRAN with its Direct Matrix Abstraction Program (DMAP) language creates the reduced flexbody models and transformation matrices.
- Dynasty, by Caterpillar Inc., is used for the flexbody simulation.
- The detailed nonlinear breakout model is built for ABAQUS.
- An in-house program, POASI, adds the forces and imposes the motions to the ABAQUS model.
- Note: The 'ASI' in PRASI and POASI comes from the acronym of the internal process name we call 'Advanced Structural Interface'

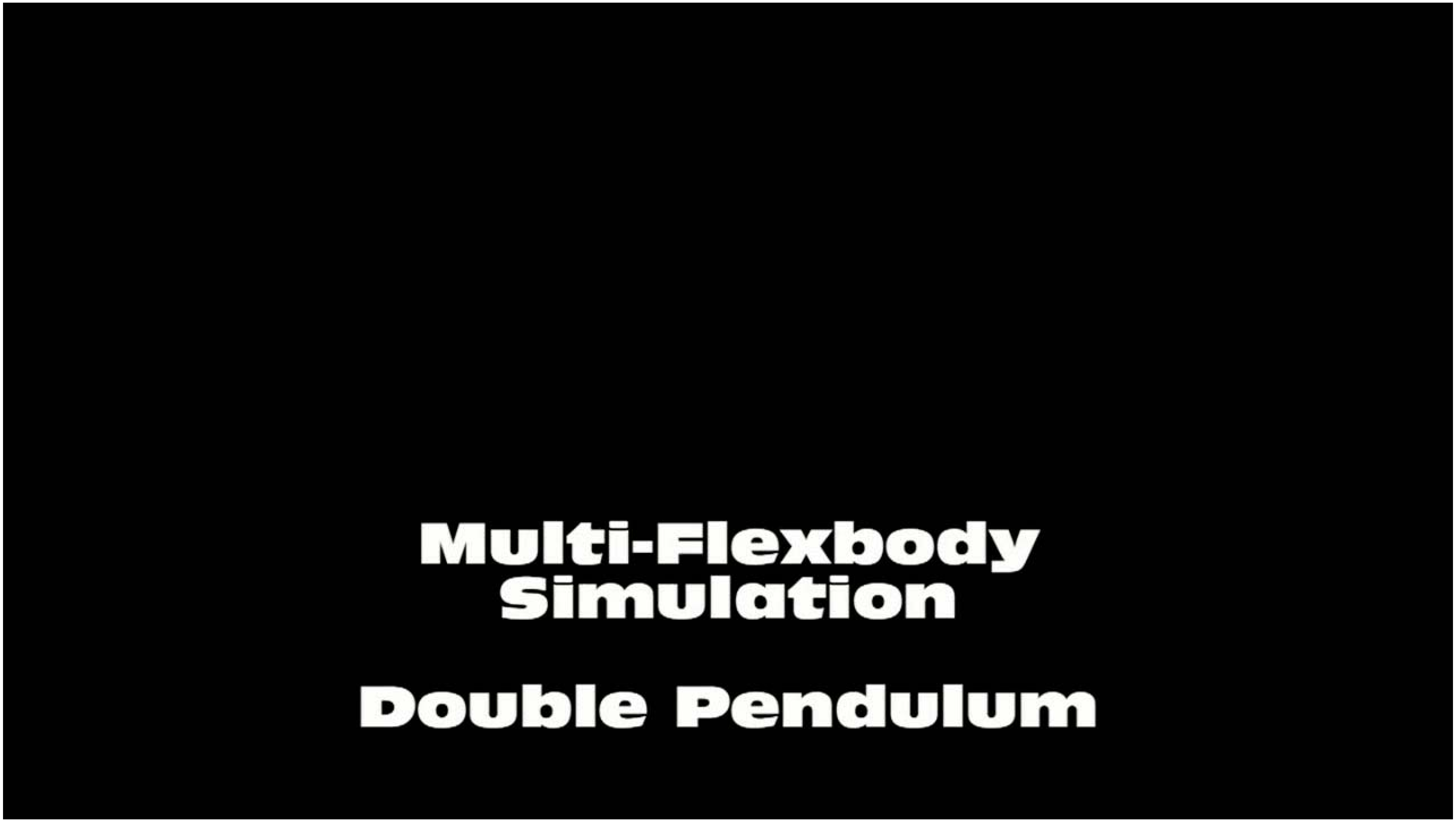
# Process Flow Chart



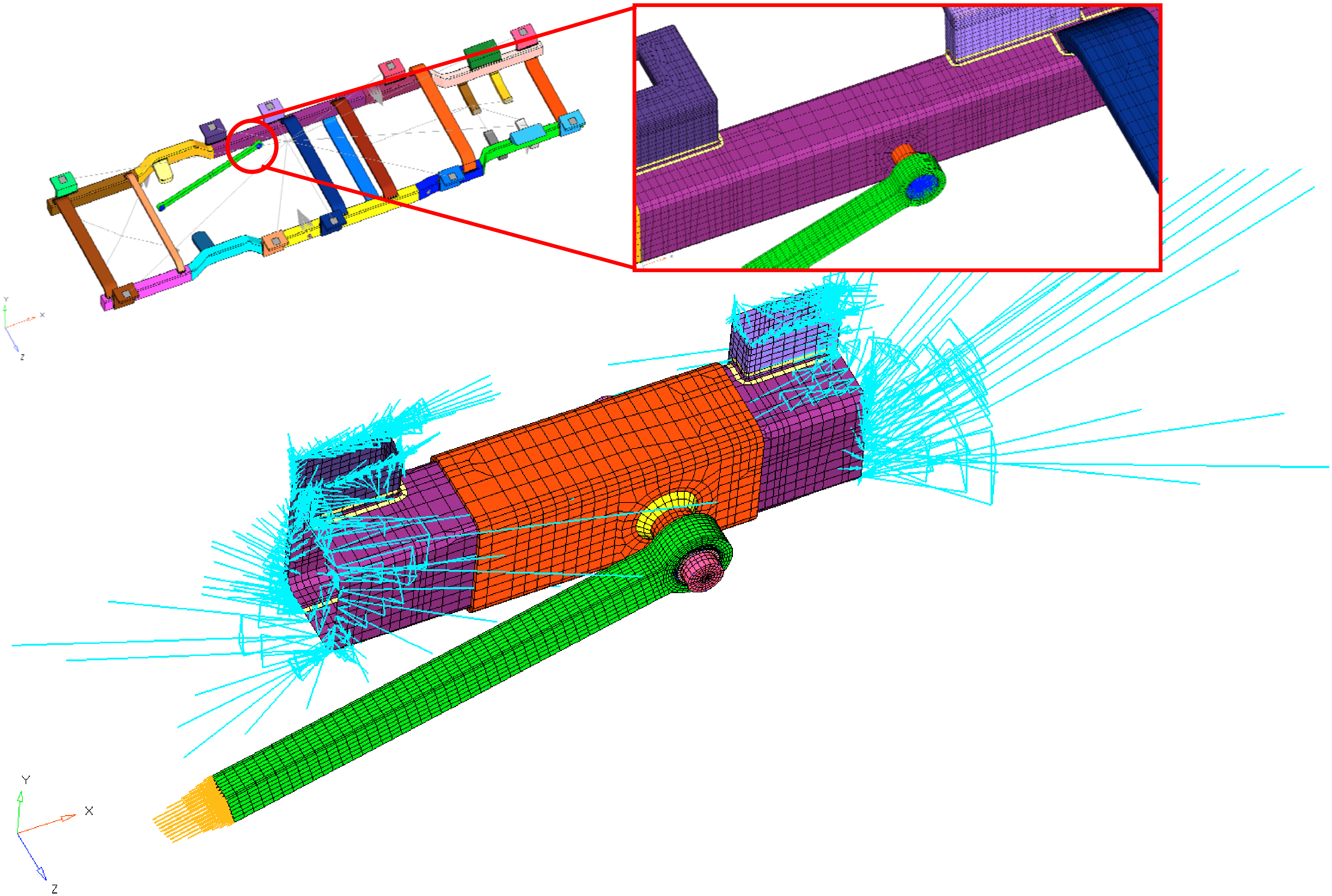
# Academic Model, Double Pendulum



# Academic Model, Double Pendulum



# Application Model, SUV Linkage



# Application Model, SUV Linkage

**Multi-Flexbody Simulation**

**Vehicle Backing into Berm**

# Questions / Discussion

- Thank you for your attention.

- Contact Information:

Paul D Fotsch
VPDS Inc
paul.fotsch@vpds-inc.com