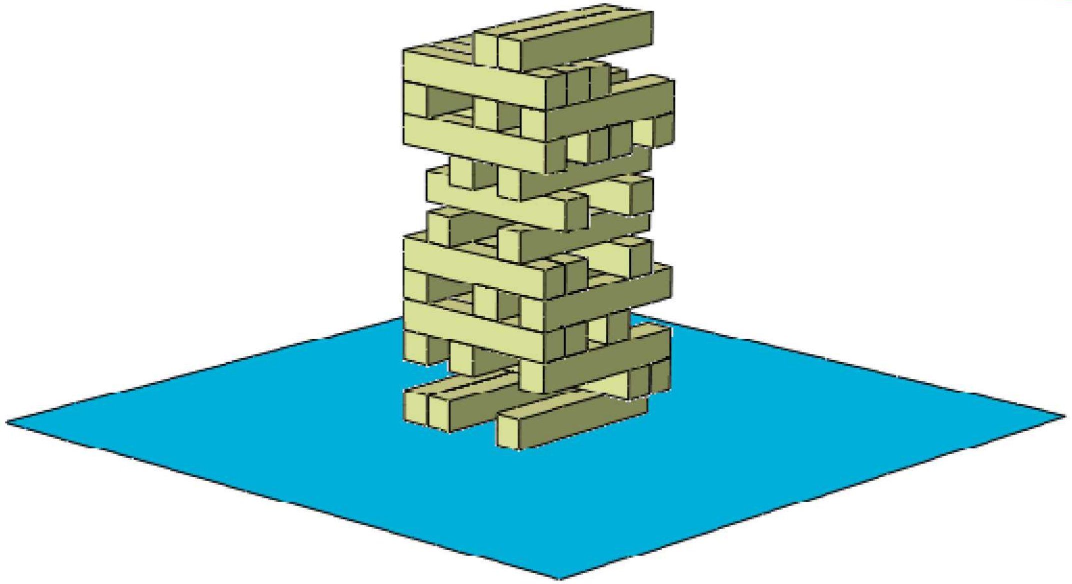
SAND2020-2194C

Substructuring in ABAQUS



Ben Moldenhauer[1], Matt Allen[1],

Dan Roettgen[2] & Brian Owens[2]

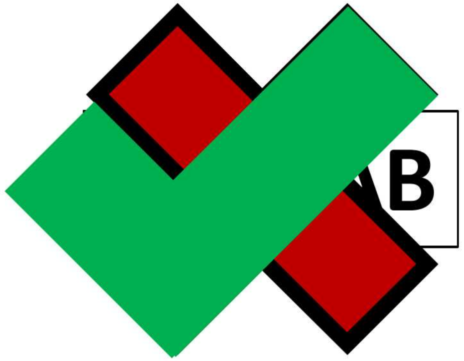
(1) University of Wisconsin – Madison

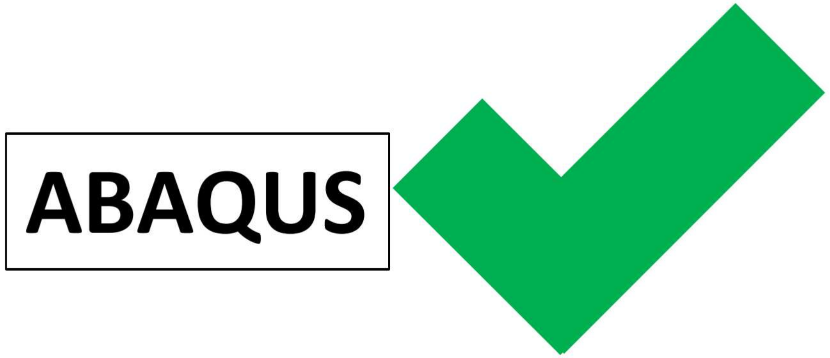
(2) Sandia National Labs

Sandia National Laboratories is a multimission laboratory managed and operated by National Technology and Engineering Solutions of Sandia, LLC., a wholly owned subsidiary of Honeywell International, Inc., for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA-0003525.

Motivation

- Usual substructuring procedure involves importing FEM results into MATLAB to do all calculations
  - Bookkeeping can be cumbersome, and yields results with limited usability

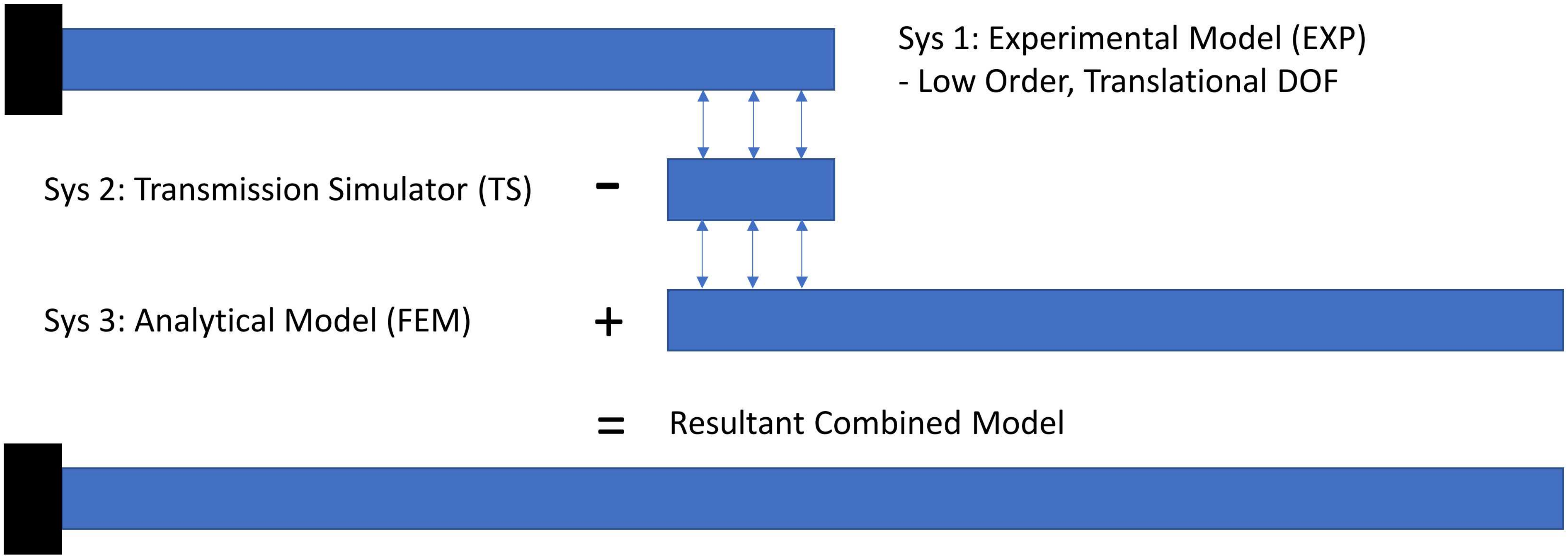




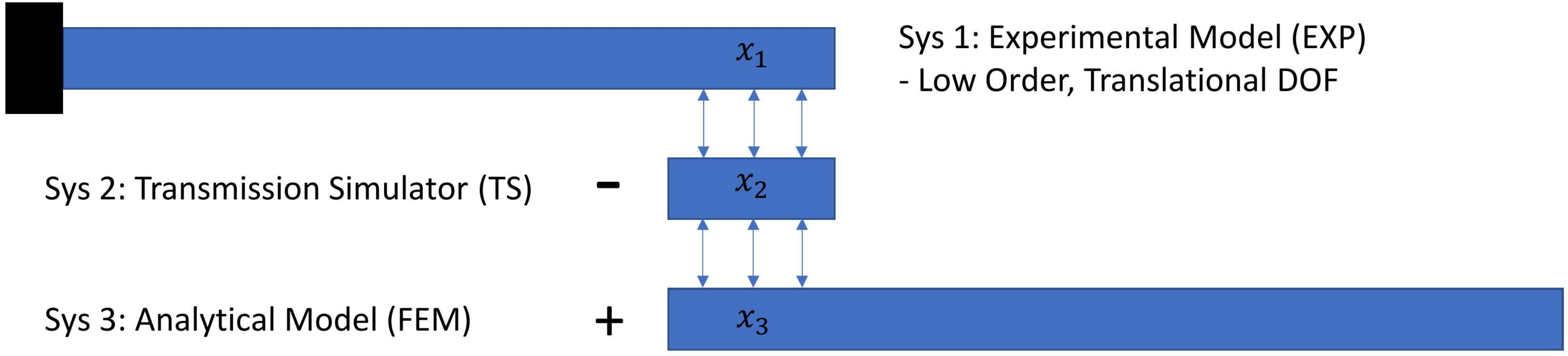
- Perform substructuring calculations during ABAQUS modal solution
  - Allows for much greater post processing options, i.e. ‘Full Field’ Results - Stress, Strain, Displacement, etc.

Substructuring Theory – Transmission Simulator Method

- Substructuring techniques allow for the dynamics of several subcomponents to be joined, approximating the assembly dynamics



Transmission Simulator Math

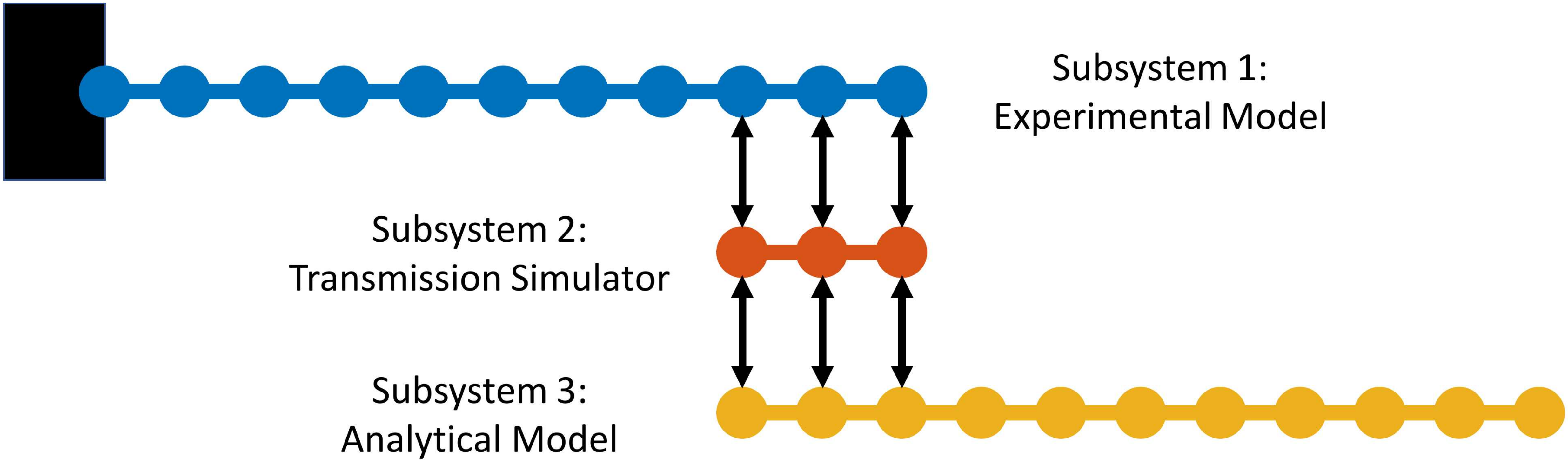


$$\begin{bmatrix} -1 & 1 & 0 \\ 0 & 1 & -1 \end{bmatrix} \begin{bmatrix} \{x_1\} \\ \{x_2\} \\ \{x_3\} \end{bmatrix} = [0] \rightarrow \begin{bmatrix} -\Phi_1 & \Phi_2 & 0 \\ 0 & \Phi_2 & -\Phi_3 \end{bmatrix} \begin{bmatrix} \{q_1\} \\ \{q_2\} \\ \{q_3\} \end{bmatrix} = [0] \rightarrow \begin{bmatrix} \Phi_2^\dagger & 0 \\ 0 & \Phi_2^\dagger \end{bmatrix} \begin{bmatrix} -\Phi_1 & \Phi_2 & 0 \\ 0 & \Phi_2 & -\Phi_3 \end{bmatrix} \begin{bmatrix} \{q_1\} \\ \{q_2\} \\ \{q_3\} \end{bmatrix} = [0]$$

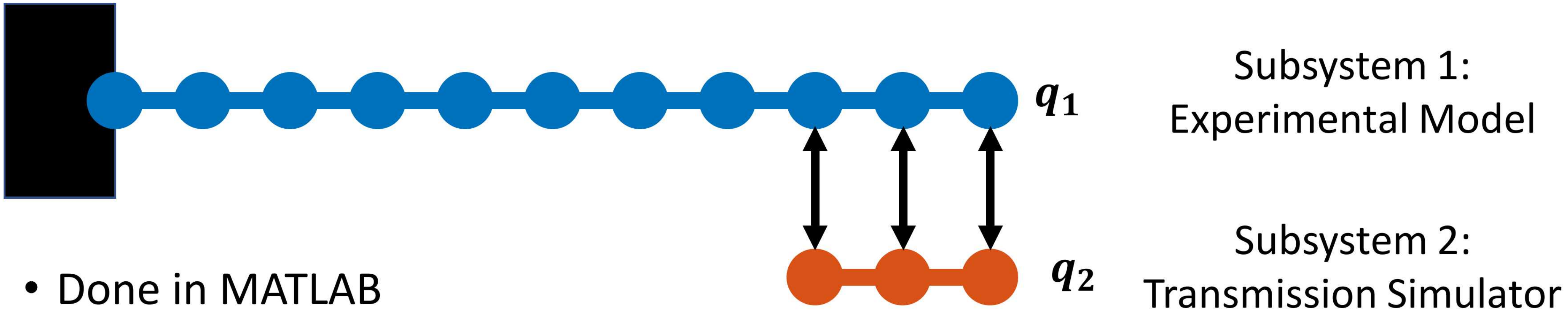
$$\rightarrow \begin{bmatrix} -\Phi_2^\dagger\Phi_1 & \Phi_2^\dagger\Phi_2 & 0 \\ 0 & \Phi_2^\dagger\Phi_2 & -\Phi_2^\dagger\Phi_3 \end{bmatrix} \begin{bmatrix} \{q_1\} \\ \{q_2\} \\ \{q_3\} \end{bmatrix} = [0] \rightarrow \begin{bmatrix} -\Phi_2^\dagger\Phi_1 & I & 0 \\ 0 & I & -\Phi_2^\dagger\Phi_3 \end{bmatrix} \begin{bmatrix} \{q_1\} \\ \{q_2\} \\ \{q_3\} \end{bmatrix} = [0]$$

$$Bq = 0$$
$$null(B) = L$$
$$\widehat{M} = L^T M L\, ,\, \widehat{K} = L^T K L$$
$$[\Phi, \lambda] = eig(\widehat{K}, \widehat{M})$$

Analytical Test Case & Method Procedure



Uncouple TS From Cantilever Beam



- Done in MATLAB
- Translation DOF constrained
- Constrained M and K can be "corrected" to be positive definite and yield real Eigen solutions

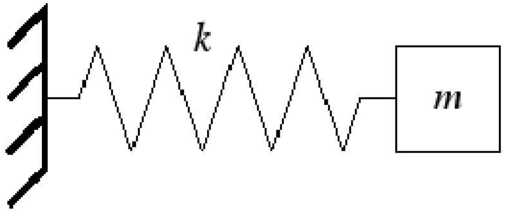
$$\begin{bmatrix} -\Phi_2^{\dagger}\Phi_1 & I \end{bmatrix} \begin{bmatrix} q_1 \\ q_2 \end{bmatrix} = [0]$$

$$\rightarrow \boxed{\Phi_{\text{Uncoupled}} \; \& \; \omega_{n_{\text{Uncoupled}}}}$$

These must be real valued to import the modal model into ABAQUS

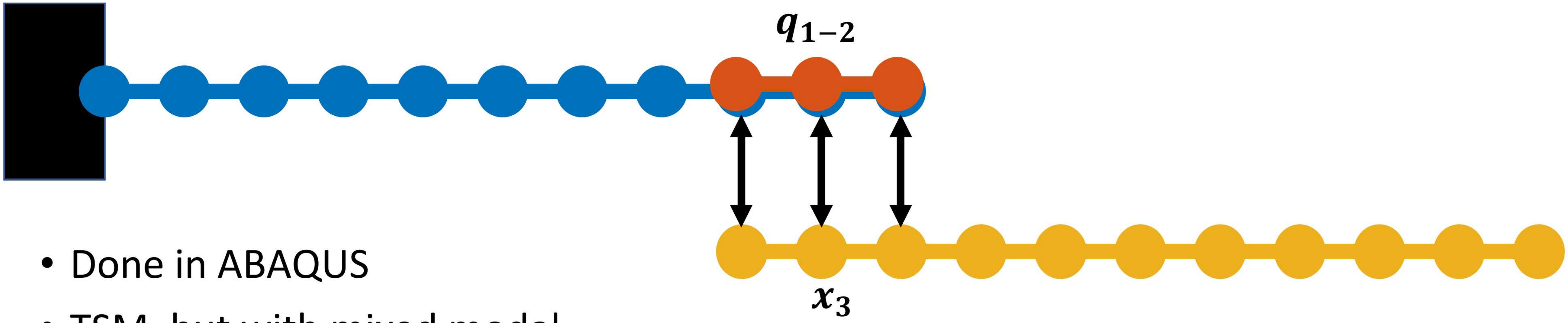
Importing Uncoupled Model into ABAQUS

- MATLAB script writes an auxiliary input file that:
  - Generates nodes for each modal DOF q_{1-2}
    - Assign unit point mass $m = 1$
    - Attach grounded spring with $k = \omega_{n_{1-2}}^2$
    - Constrain DOF 2-6



  - Constraints: Linear Equation Multi-Point Constraints
    - Equation terms defined by: $\begin{bmatrix} -\Phi_2^{\dagger}\Phi_{1-2} & \Phi_2^{\dagger} \end{bmatrix} \begin{bmatrix} q_{1-2} \\ x_3 \end{bmatrix} = [0]$
    - Must put equations in RREF - ABAQUS eliminates first DOF in each EQ, so it can only appear once

Couple Analytical Subsystem onto Previous Result

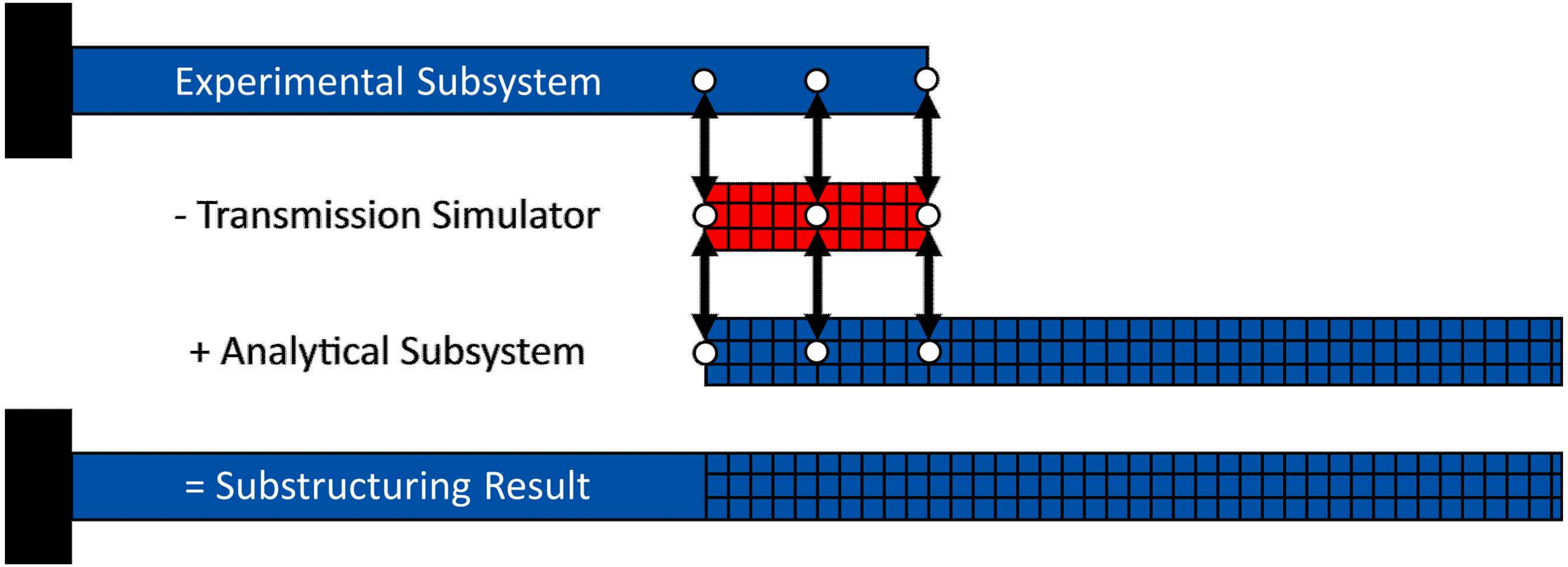


- Done in ABAQUS
- TSM, but with mixed modal and physical coordinates
- Yields prediction of resultant combined beam (a longer cantilever beam)

$$\begin{bmatrix} -\Phi_2^{\dagger}\Phi_{\text{Uncoupled}} & \Phi_2^{\dagger} \end{bmatrix} \begin{bmatrix} q_{\text{Uncoupled}} \\ x_3 \end{bmatrix} = [0]$$

$$\rightarrow \boxed{\Phi_{\text{Coupled}} \;\&\; \omega_{n_{\text{Coupled}}}}$$

ABAQUS Beam Models

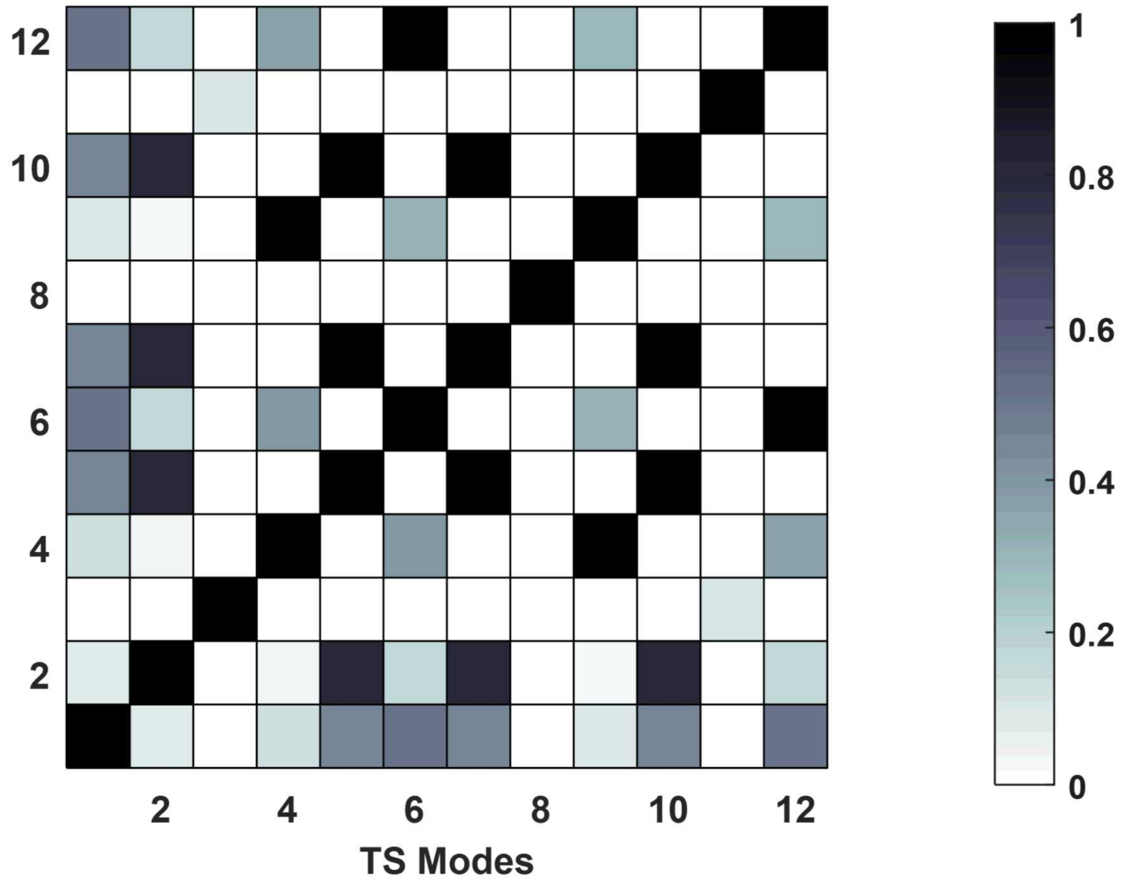


- 2 Node Linear Beam Elements (B21)
- Aluminum properties:
  - 2700 [kg/m^3], .33 [-], 70e9 [GPa]
  - 1 [cm^2] Cross Section

- EXP: 1 [m], 500 Elements, cantilever
- TS: 0.3 [m], 500 Elements, free-free
- FEM: 1 [m], 500 Elements, free-free
- Truth: 1.8 [m], 900 Elements, cantilever

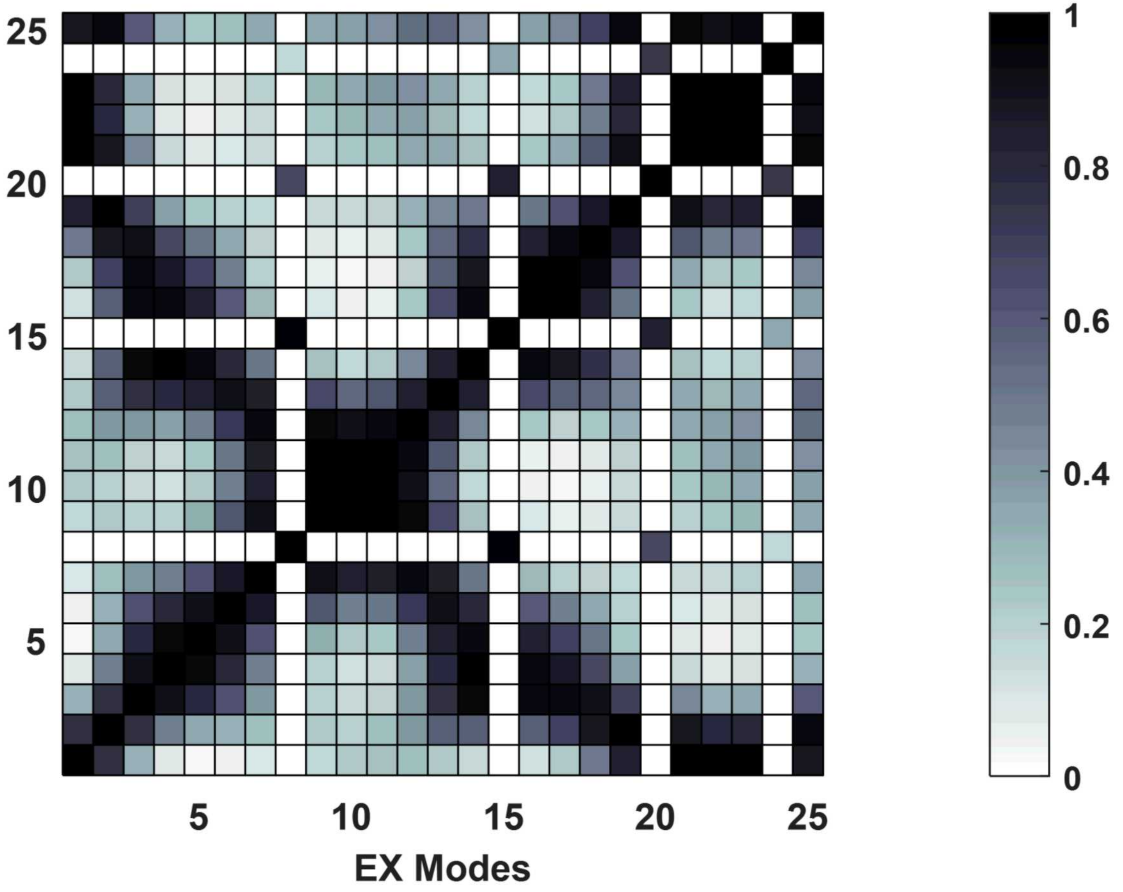
Beam Results: Subsystem Modes Used

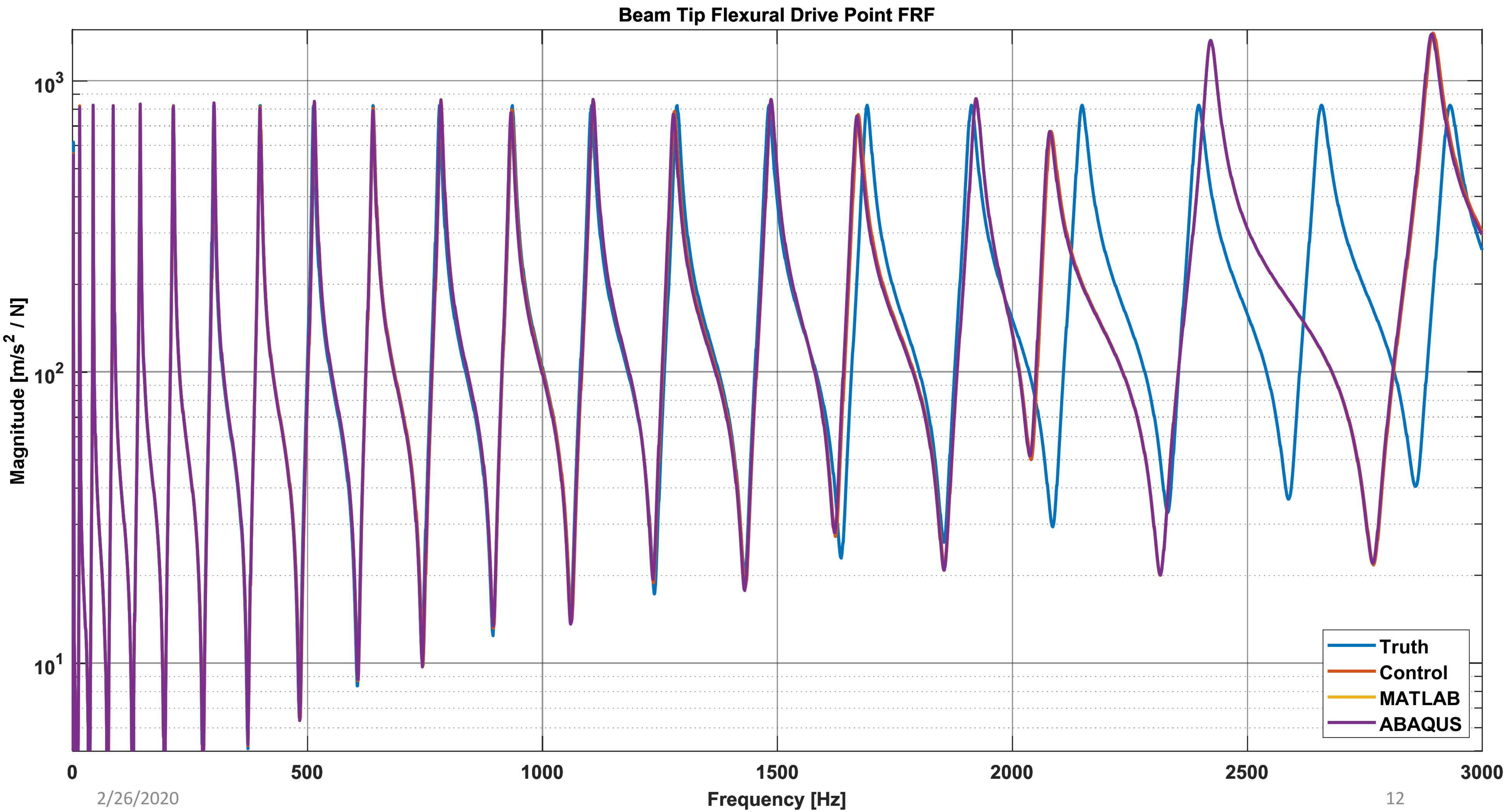
- Experimental Model:
  - 10 Flexural Modes - 2050 Hz
  - 4 Axial Modes – 8910 Hz
- Transmission Simulator:
  - 3 RBMs
  - 1 Flexural Mode – 1297 Hz
  - 1 Axial Mode – 12730 Hz
- Analytical Model: All Modes



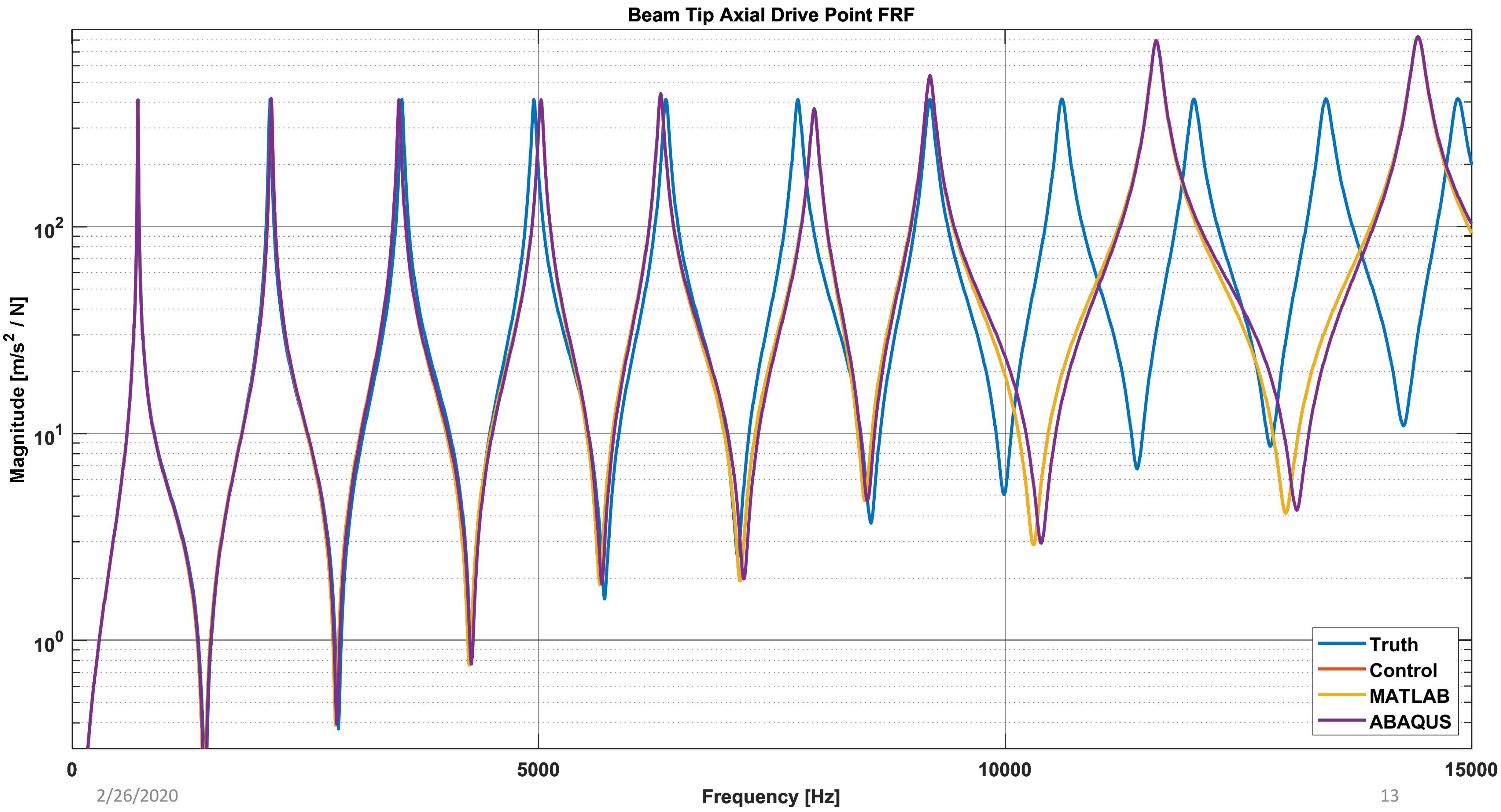
Beam Results: Subsystem Modes Used

- Experimental Model:
  - 10 Flexural Modes - 2050 Hz
  - 4 Axial Modes – 8910 Hz
- Transmission Simulator:
  - 3 RBMs
  - 1 Flexural Mode – 1297 Hz
  - 1 Axial Mode – 12730 Hz
- Analytical Model: All Modes





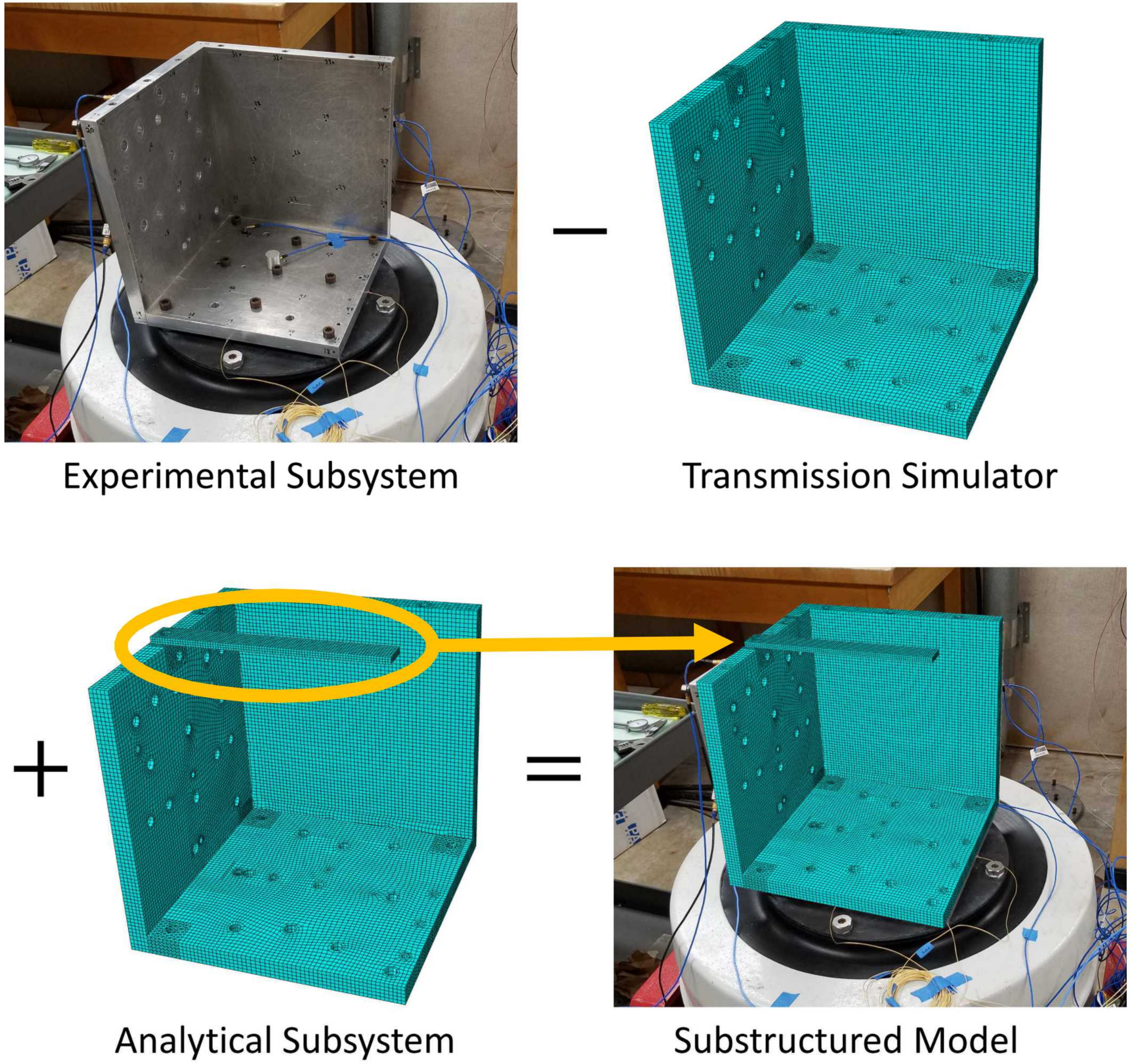
Beam Tip Flexural Drive Point FRF
Magnitude [m/s^2 / N]
10^3
10^2
10^1
0
500
1000
1500
2000
2500
3000
Frequency [Hz]
Truth
Control
MATLAB
ABAQUS



Beam Tip Axial Drive Point FRF
Magnitude [m/s^2 / N]
10^2
10^1
10^0
0
5000
10000
15000
Frequency [Hz]
Truth
Control
MATLAB
ABAQUS

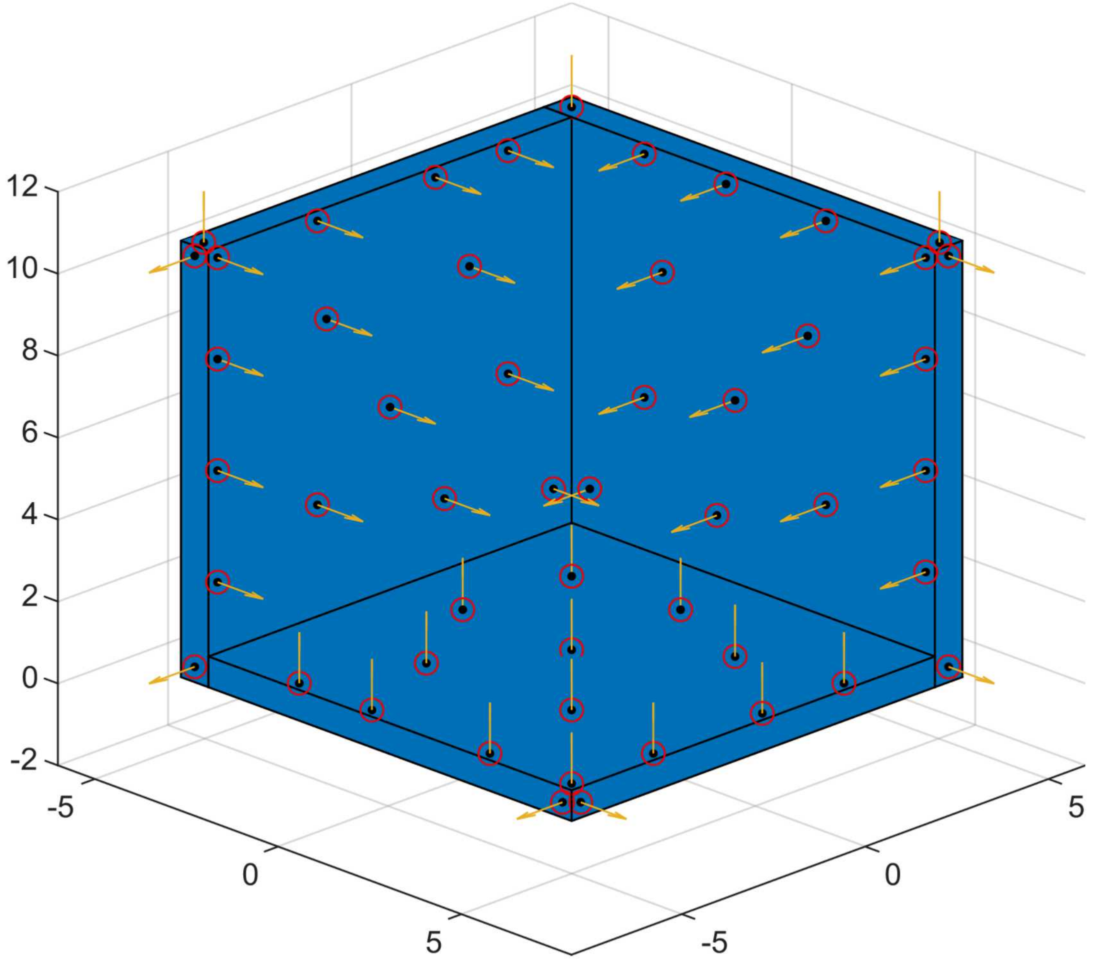
Half Cube Test Case

- Experimental data and FEMs from project looking at attaching fixtures to a shaker
- Challenging scenario based on experimental data and a dynamically complex structure
- Experimental Truth data is available to compare with

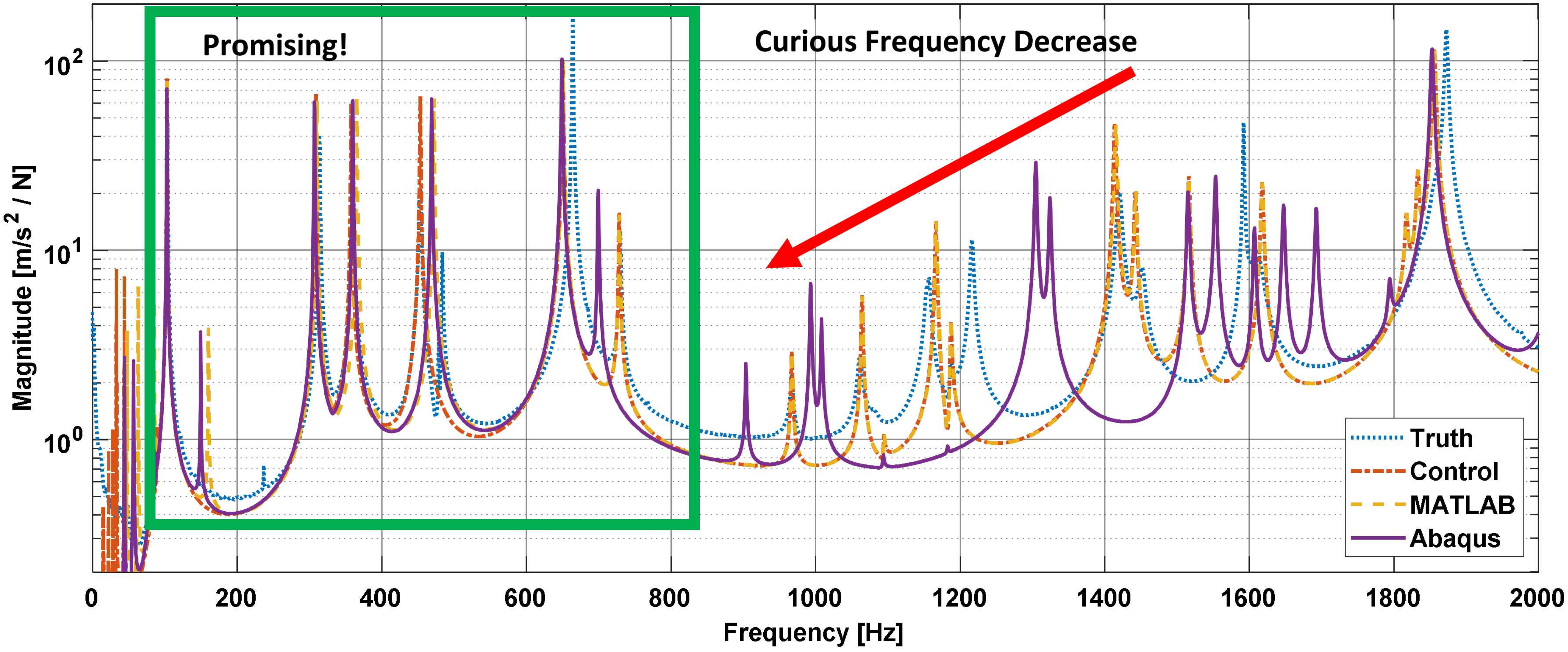


Mode Selection for the Subsystems

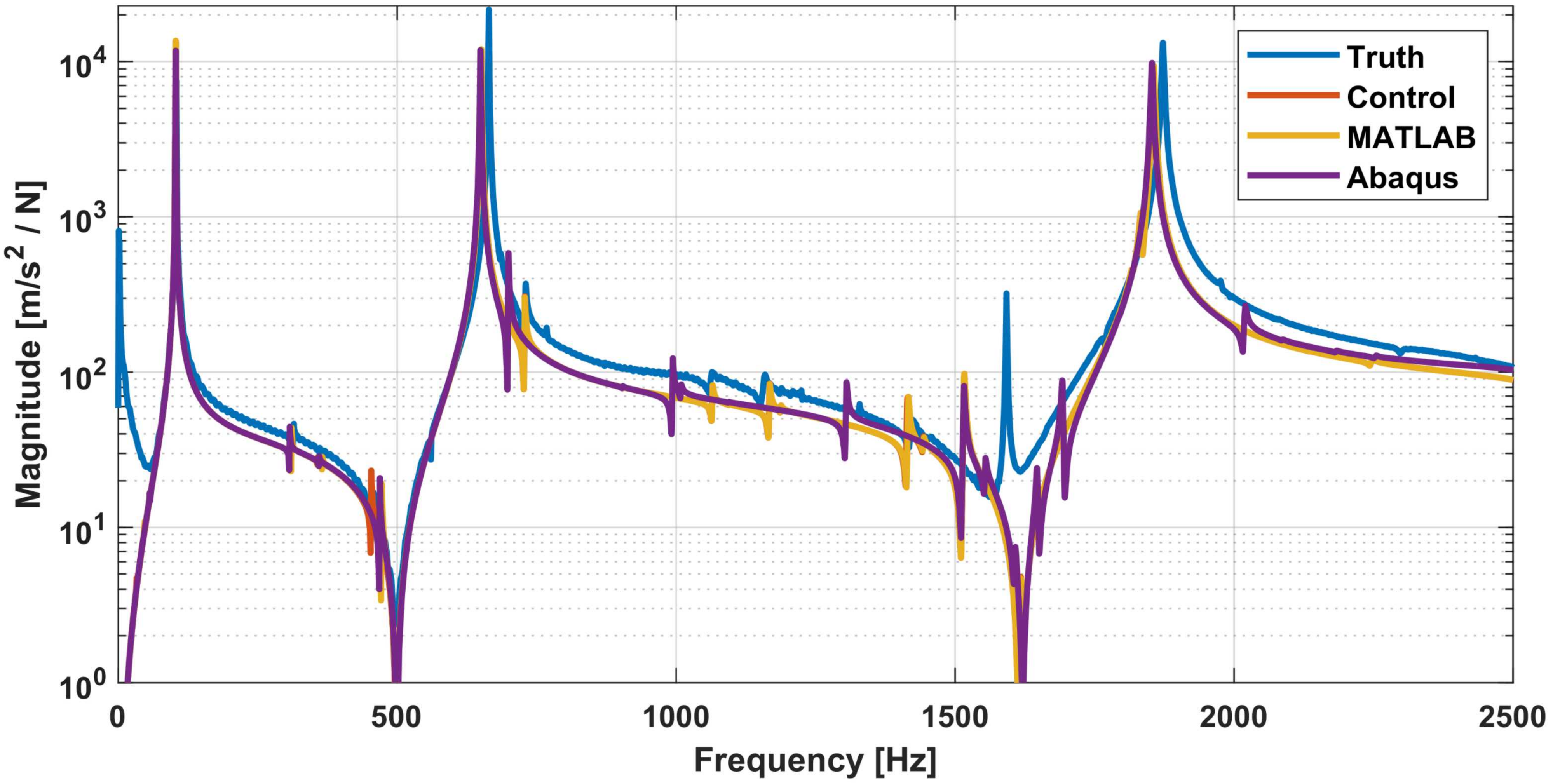
- 51 DOF used in experiment
- Experimental: 32 modes up to 3400 Hz
- TS: 22 modes up to 3800 Hz
- Analytical: All (100 modes up to 12200 Hz when imported into MATLAB)



Truth Data vs Substructuring Predictions

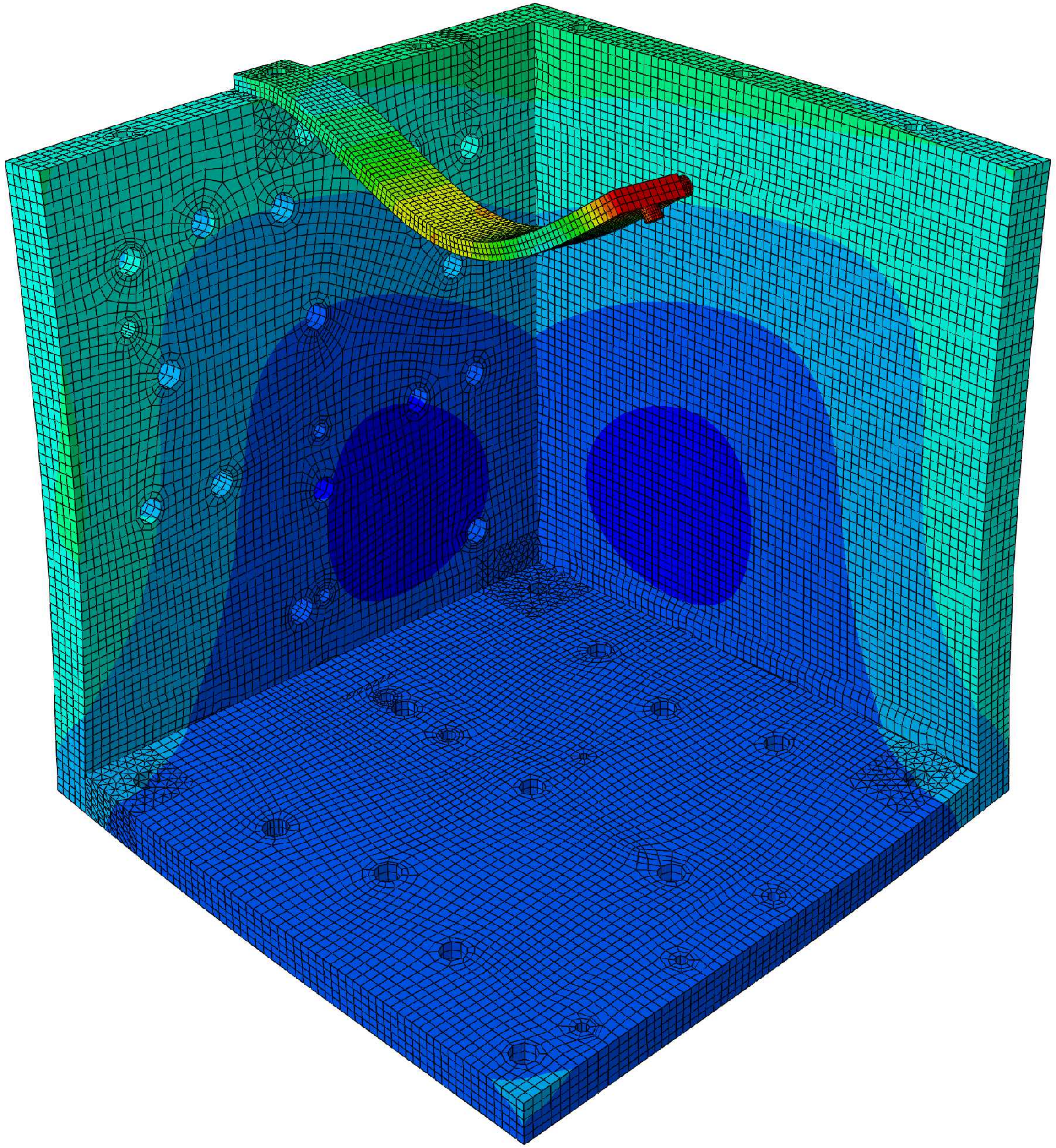


Beam Tip FRF Accurate For All Methods



Abaqus Produces Full FEM Results

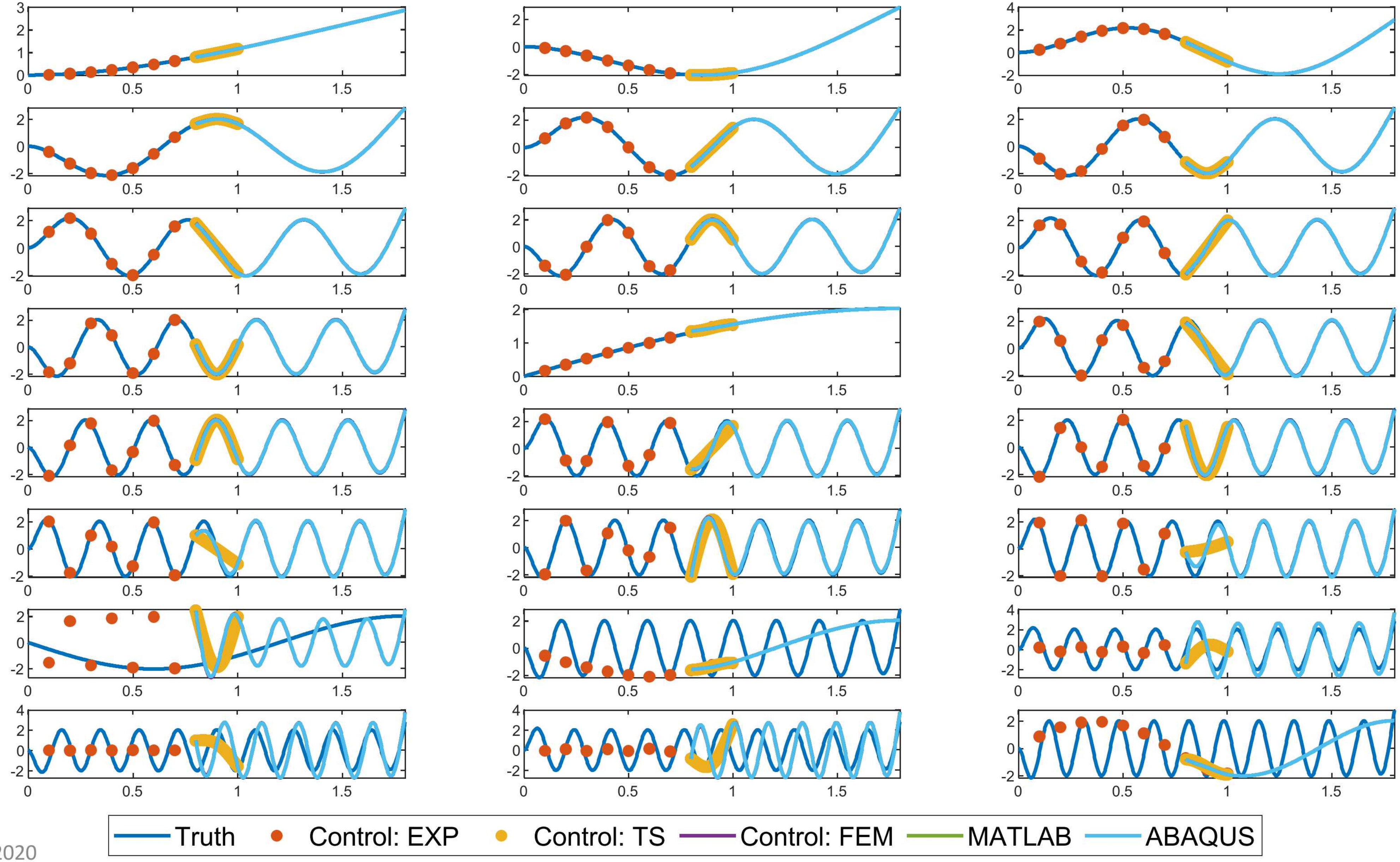
- Mode near 700 Hz is shown
- Participation of close beam mode is visible
- Bottom is correctly constrained



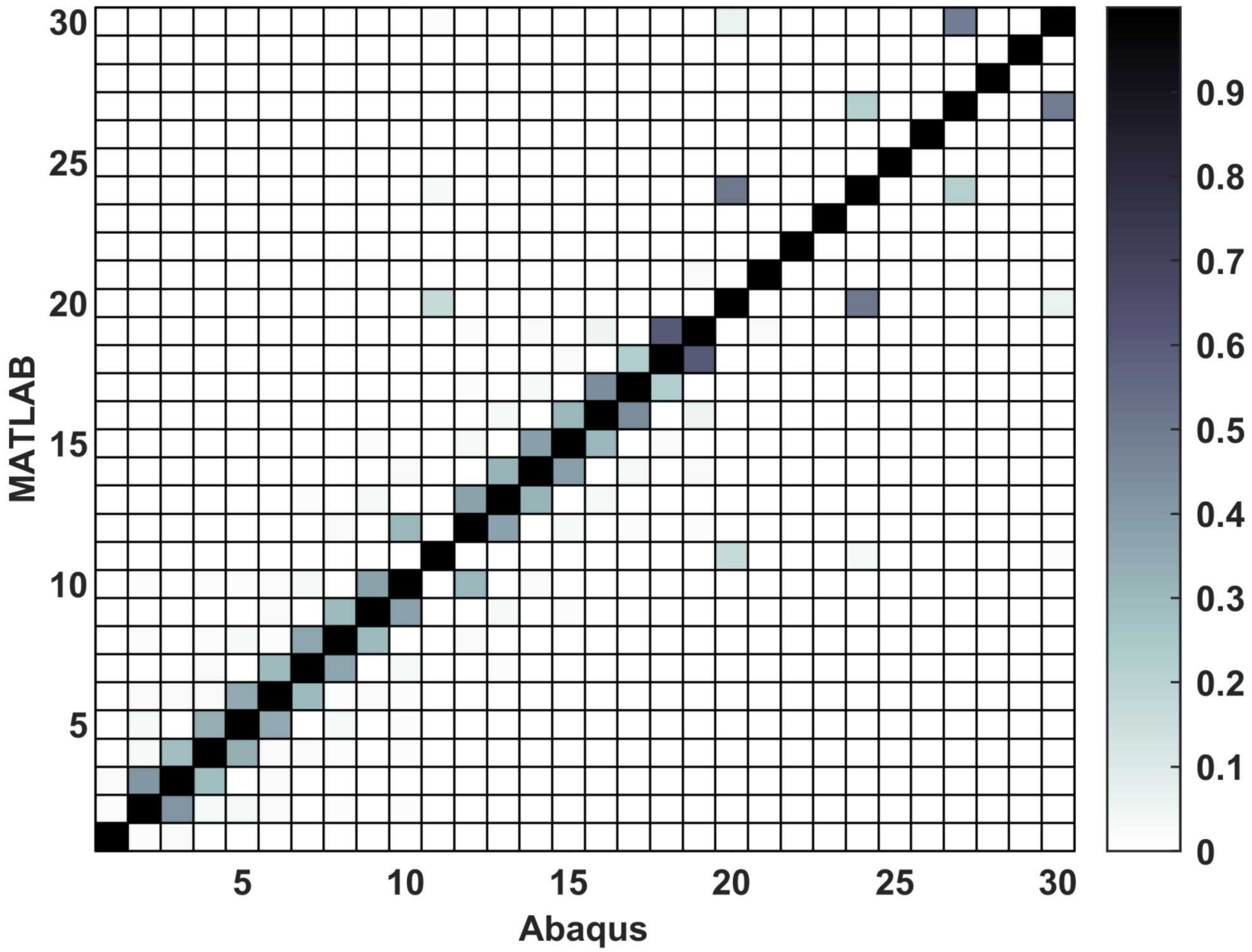
Conclusions

- Substructuring calculations can be performed in Abaqus
  - Uncouple TS from EXP in MATLAB, then import into Abaqus and apply Linear Eq MPCs
  - Decoupling step can be problematic and seems to always degrade results
- Beam numerical case study yielded good results in an ideal scenario
  - Near perfect agreement between MATLAB and Abaqus results
  - Accurate to truth data through full input frequency range
- Half cube experimental test case is a challenging, real-world application
  - Resulting Abaqus assembled model is accurate, but over a limited frequency range when compared to what can be computed in MATLAB
  - Abaqus result seems ‘weighed down’ by the coupling of every analytical subsystem mode – MATLAB results are better if a truncated set is used.
  - Abaqus solution allows for the assembled model to be easily post-processed in the FEA software i.e. full FEM deformation and stress/strain analysis

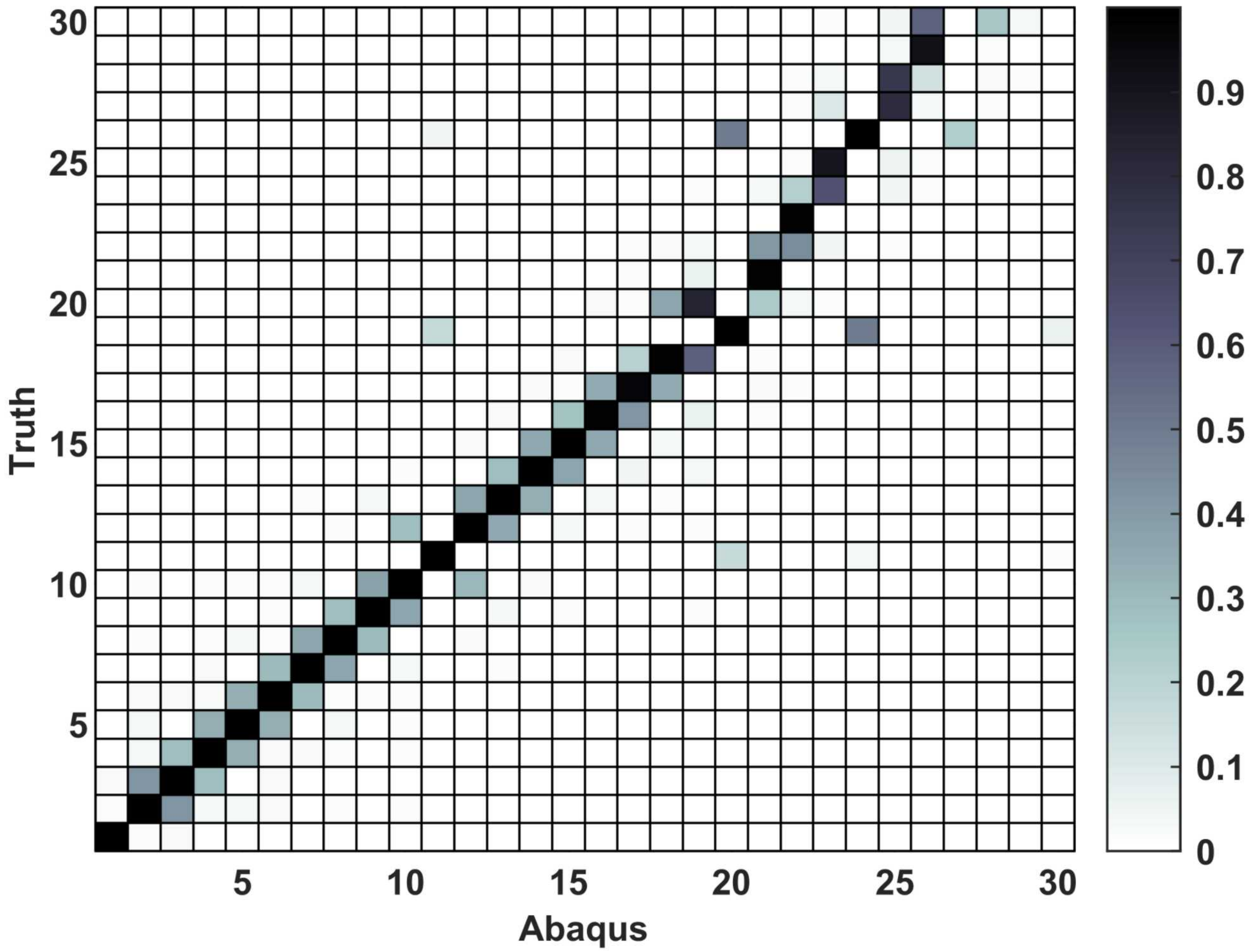
Modeshapes



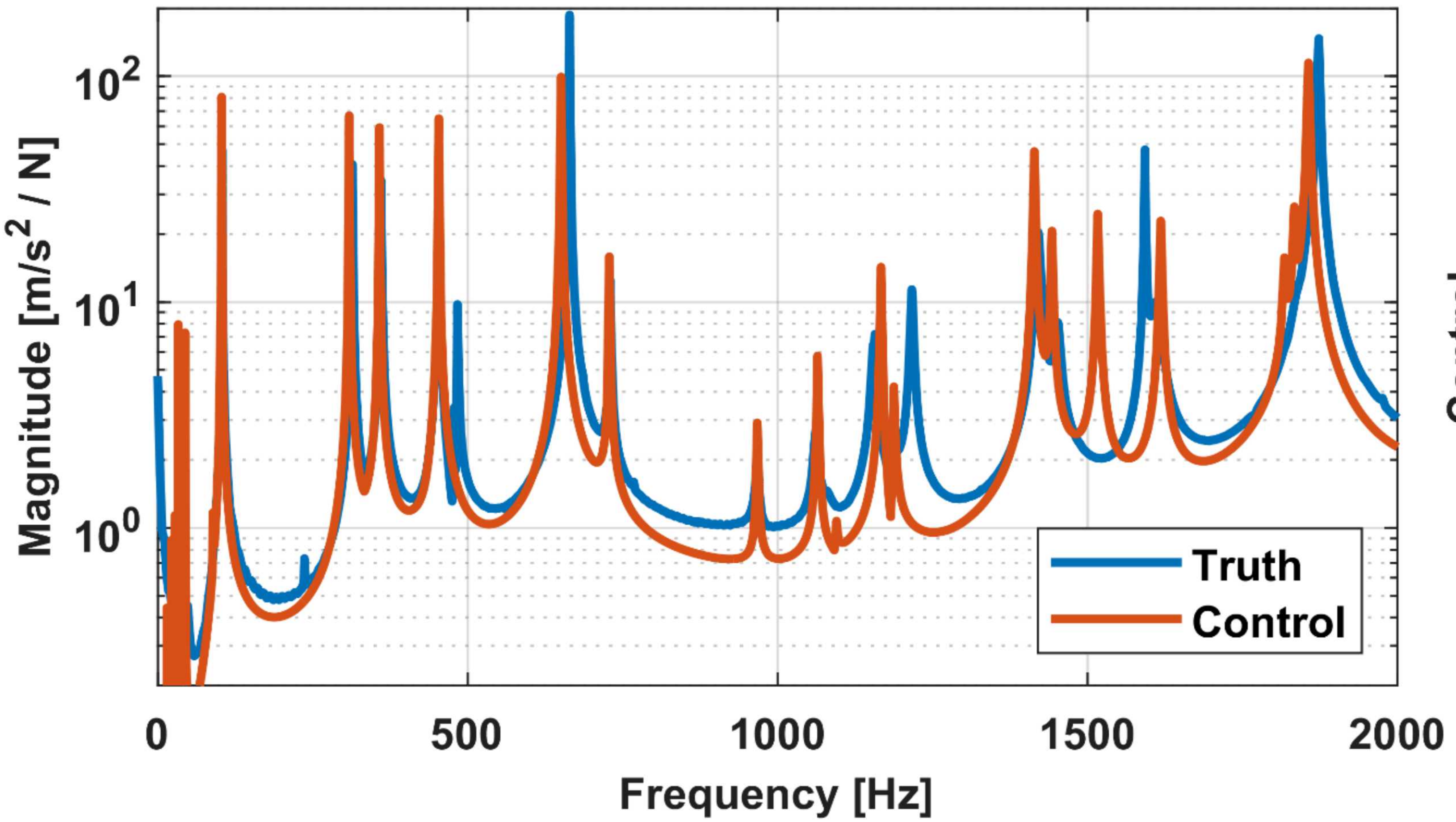
MAC: ABAQUS to MATLAB

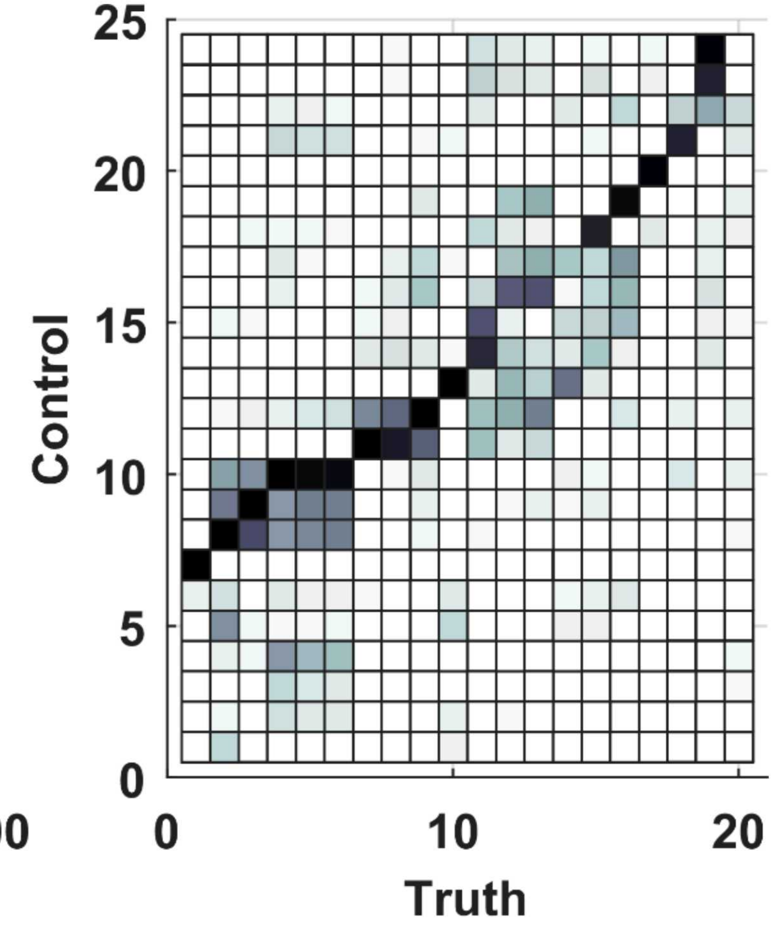


MAC: ABAQUS to Truth

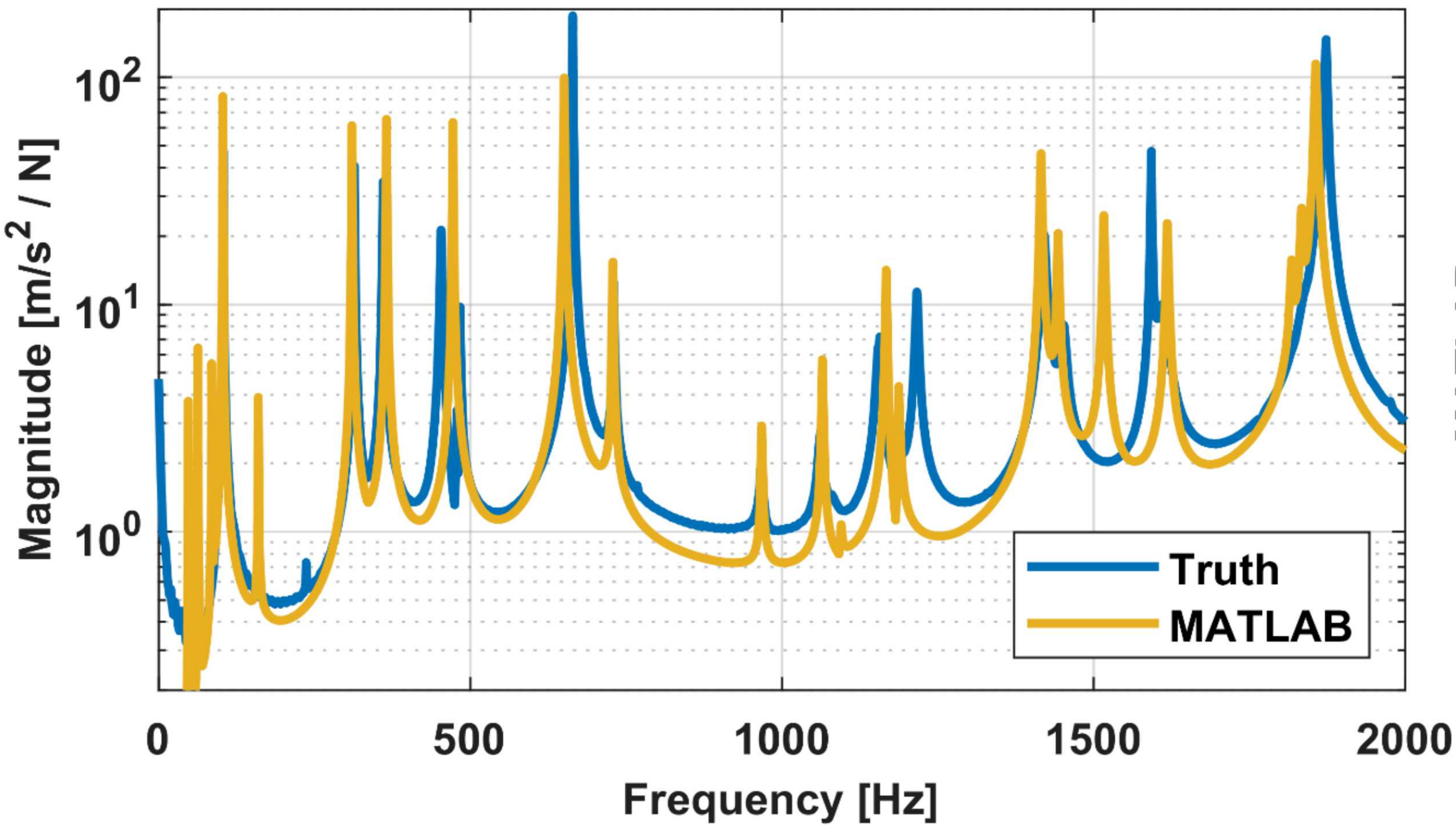


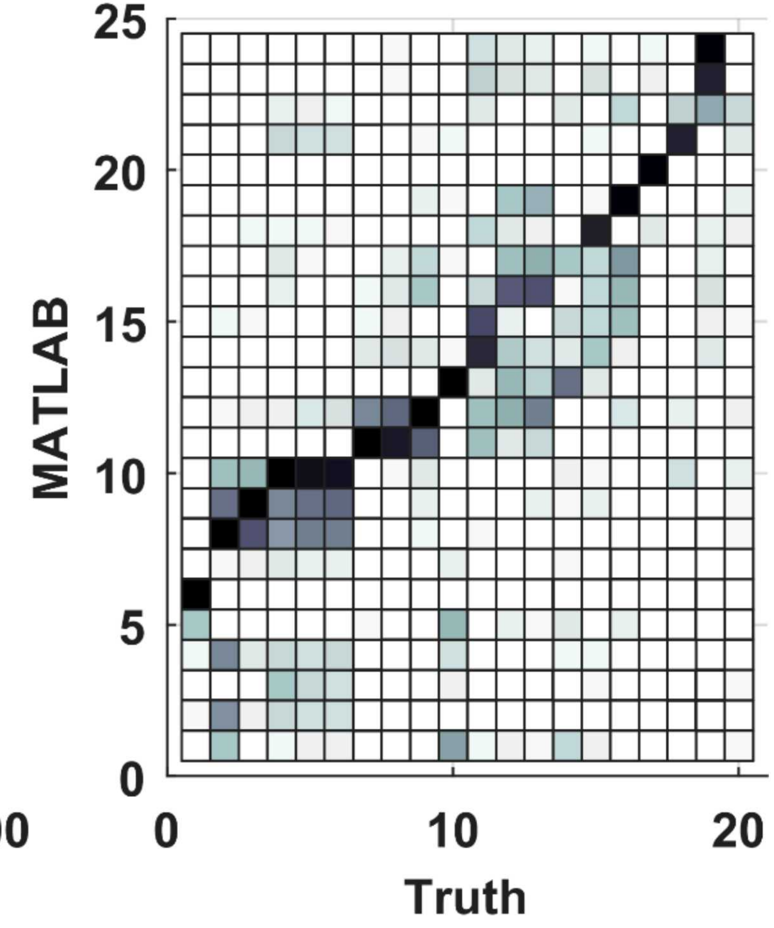
Results: Truth Data vs Standard MATLAB





Results: Truth Data vs Hybrid MATLAB





Results: Truth Data vs Abaqus Output