CATIA V5 Training

Foils

Generative Dynamic Response Analysis

Version 5 Release 19
September 2008

EDU_CAT_EN_GDY_FI_V5R19

Instructor Notes:

About this course

Objectives of the course

Upon completion of this course you will be able to:

- Understand the differences between harmonic and transient analyses
- Define load and restraint excitations
- Ensure that the appropriate pre-requisites are defined for the required excitation case
- Visualize and animate 3D images of the analysis results
- Generate translation, velocity and acceleration graphs
- Export result data in Text or Excel format

Targeted audience

Structural Analysts

Prerequisites

Students attending this course should have knowledge of CATIA V5 Fundamentals, Generative Part Structural Analysis Fundamentals

0.5 day

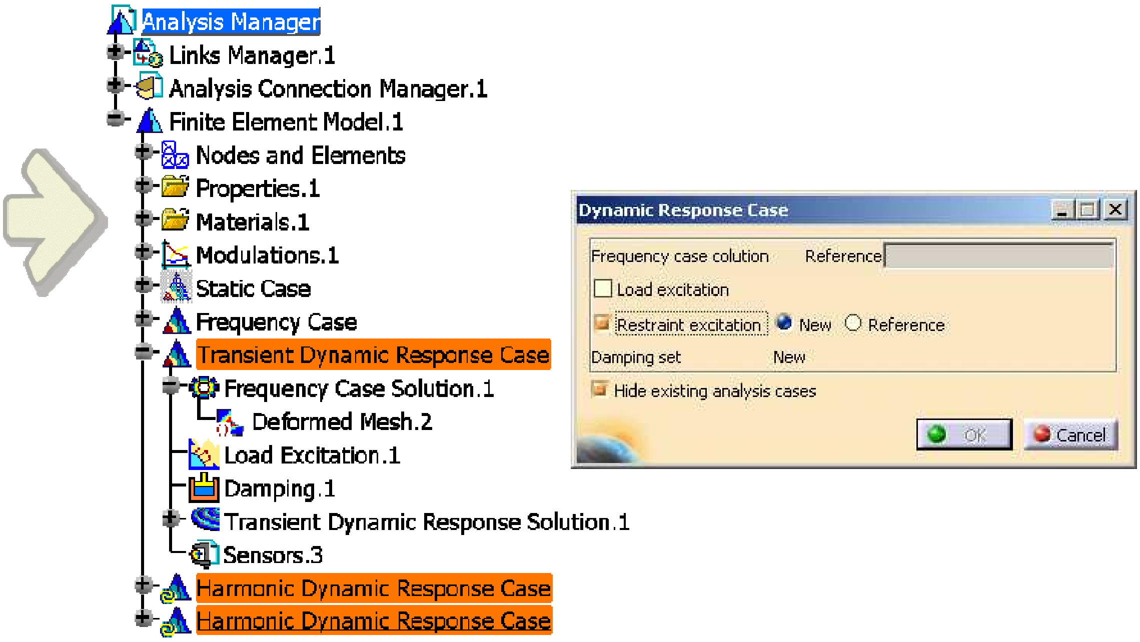
Instructor Notes:

Table of Contents

Instructor Notes:

Introduction to Dynamic Analysis

You will see the different types of dynamic analysis and what they are for



Instructor Notes:

What are the Different Types of Dynamic Analysis

A 'Dynamic Response Case' can be seen as a combination between a Static Case and a Frequency Case

When you define a Dynamic analysis, you have the choice between 2 cases:

- **Harmonic Response Analysis**
- **Transient Response Analysis**

What are they for?

- **Harmonic Response Analysis:**

  You will define a such case if the amplitude of the excitation you are applying on a part/assembly is fluctuating according to frequencies. Thus, it will allows you to answer the question: What are the constraints and displacements in a part/assembly for given frequencies?

- **Transient Response Analysis:**

  You will define a such case if the excitation you are applying on a part/assembly is fluctuating according to the Time. Thus, it allows you to answer the question: What are the constraints and displacements in a part/assembly after a given time?

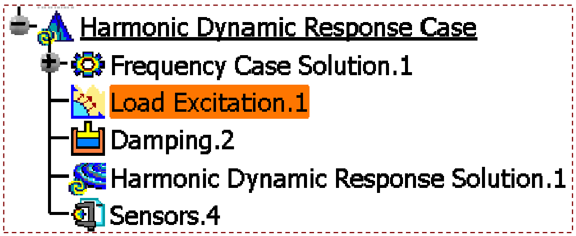
Instructor Notes:

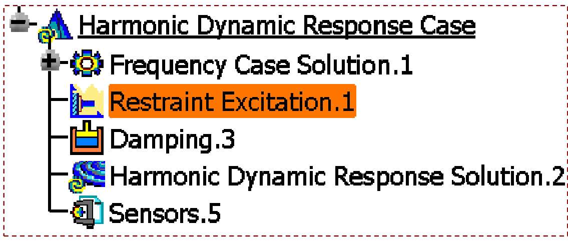
Load and Restraint Excitation

CATIA allows you to define two different types of excitation.

Once you have chosen the type of dynamic response you want, you have to specify the nature of the excitation. In both cases, you can either apply a "load" or a "restraint" excitation

- **Load Excitation Set: It allows to define a dynamic load, that will fluctuate according to the frequency or the time, depending on the dynamic case you have chosen**
- **Restraint Excitation set: It allows you to impose motion of the support, that will fluctuate according to the frequency or the time, depending on the dynamic case you have chosen**



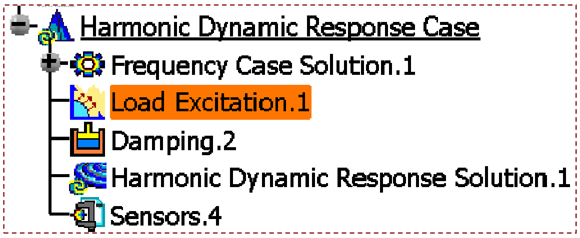


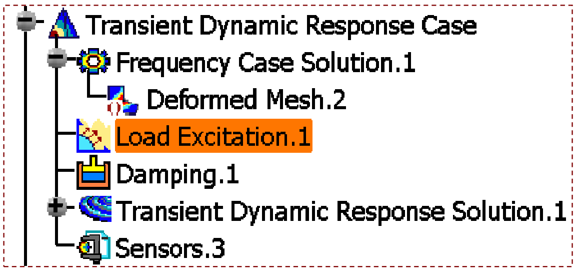
Instructor Notes:

Prerequisites

When you define a dynamic analysis for the first time, you need:

Harmonic/Transient Analysis: Load Excitation

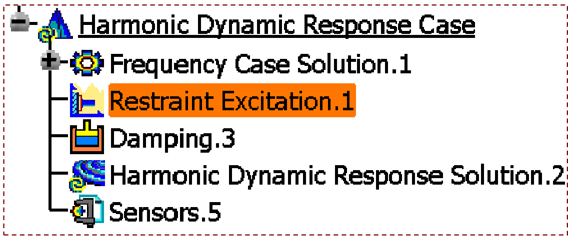


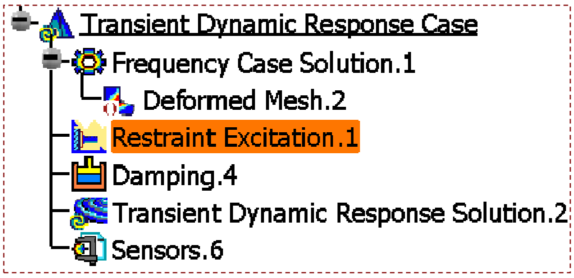


Pre-requisites

- **A computed static analysis with the Load you want to excite**
- **A computed frequency analysis that allows you to know the modal frequencies**

Harmonic/Transient Analysis: Restraint Excitation





Pre-requisites

- **A computed frequency analysis with that allows you to know the modal frequencies**

Instructor Notes:

GDY Pre-Processing

You will see how to define harmonic and transient dynamic case

- Harmonic Response Case
- Transient Response Case
- To Sum Up

Instructor Notes:

Harmonic Response Case

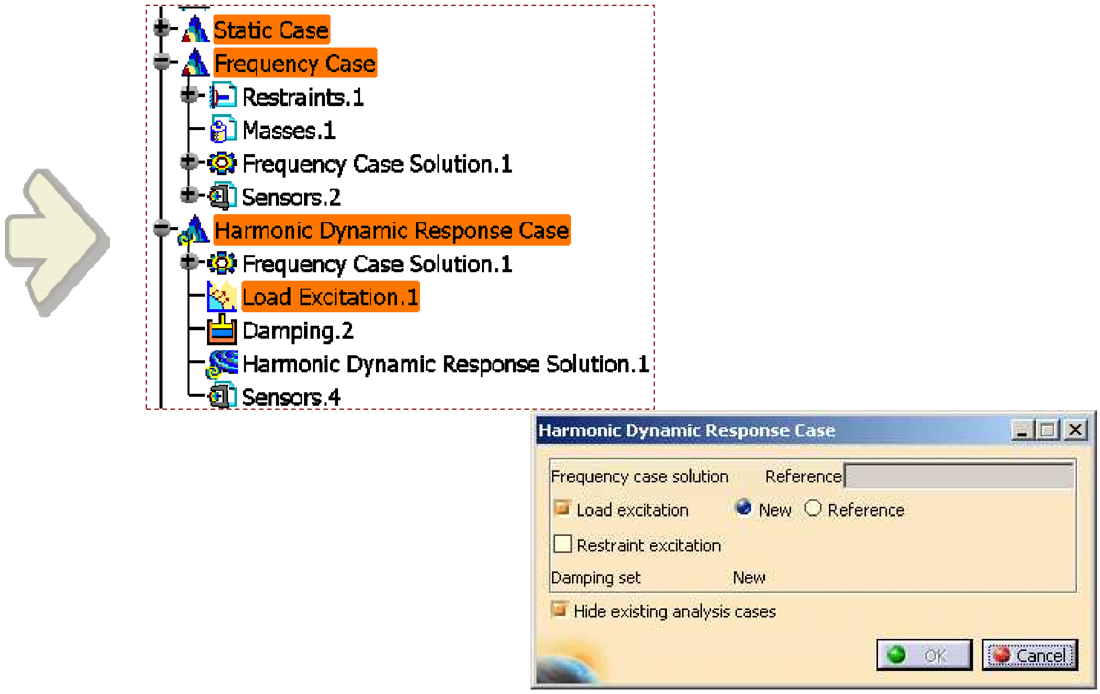
You will see different excitations cases used in Harmonic Response Analysis

- Harmonic: Load Excitation Case
- Harmonic: Restraint Excitation Case
- To Sum Up

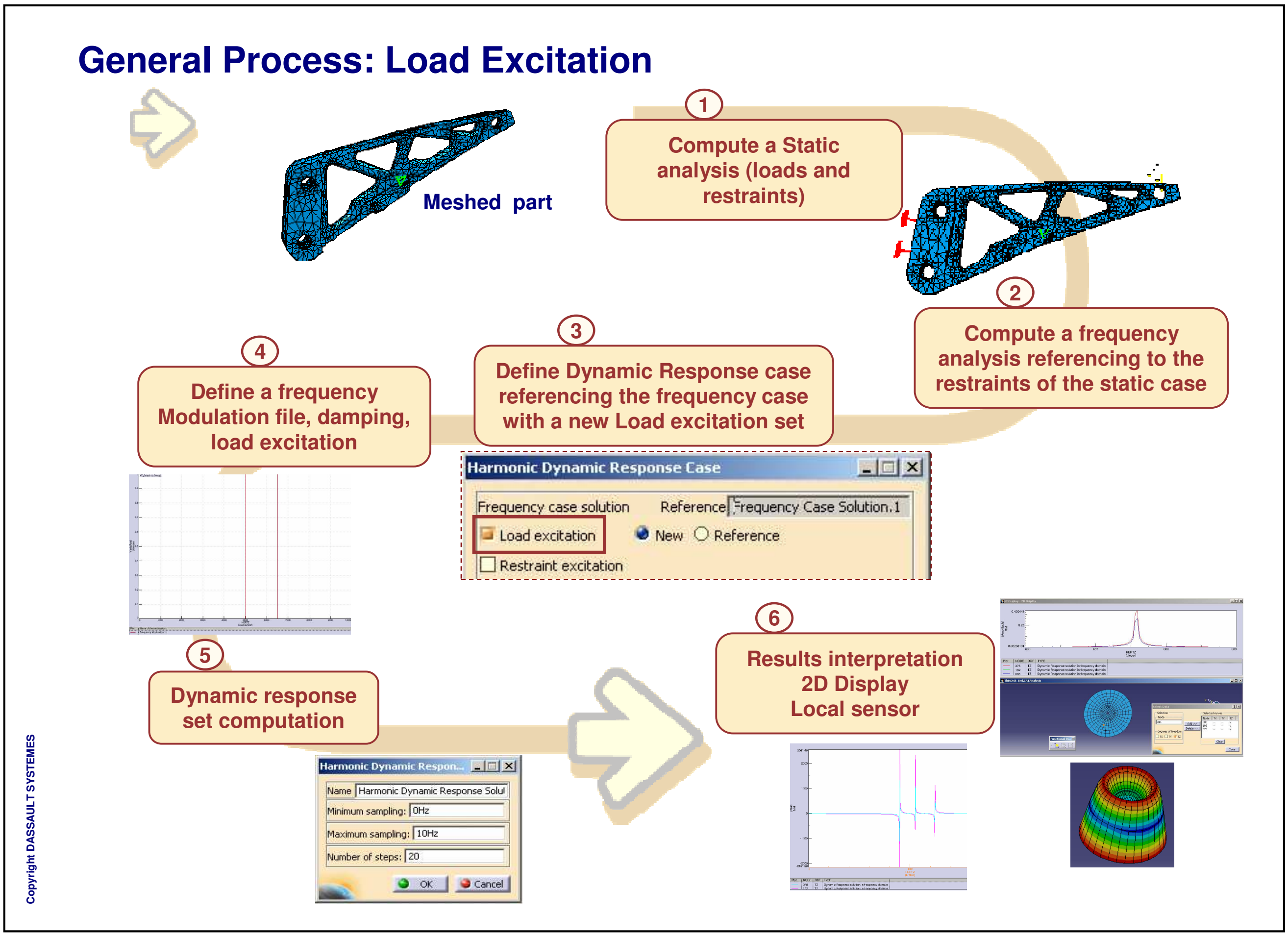
Instructor Notes:

Harmonic: Load Excitation Case

You will see how to define a Harmonic Response Case with a Load Excitation



Instructor Notes:



Instructor Notes:

- Donner des procès généraux quand cela est possible pour illustrer
- What , How, Why ne sont pas obligatoires mais doivent guider et aider dans la rédaction du foils.

Forme

- ATENTION de na pas modifier la taille du cadre intérieur de la diapo qui est dimensionne pour pouvoir imprimer en format Américain aussi bien que Européen
- Essayer de respecter la palette de couleurs proposée
- Ne pas oublier de mettre a jour le titre du cours dans le masque
- Chaque mot du titre doit commencer par une majuscule

Style:

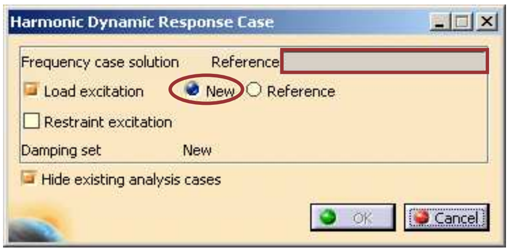
Utiliser You... A la place des phrases impersonnelles

Utiliser la voix active plutot que passive

Ecrire des phrases simples: Souvenez vous que ce document peut etre traduit en d'autres langues et donc nous devons eviter toutes ambiguites.

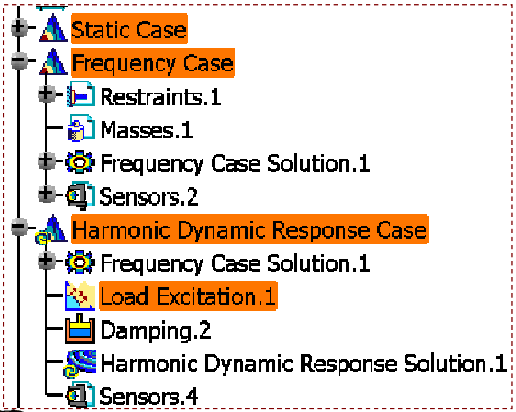
Defining a Harmonic/Load Excitation Case

When defining a ‘Dynamic Response Case’ you have to select different data.



You must refer to the pre-computed frequency analysis to take into account the modal frequencies of the system.

When you define a dynamic case for the first time, you have to check “New” load excitation. Afterwards you can either select new excitation or already created excitations as reference.



For this case, you must have computed a static and a frequency analysis.

On the left, you can see a typical tree of harmonic/load Excitation Case. Below are the main components:

First, you can see the frequency Case you have referred and the different solutions you had displayed.

Load Excitation: that is a very important thing. It will refer to the load of the pre-computed static analysis and it is on this load that you will apply a modulation. It means, you will have to define a modulation file. (cf. jobaids presented further)

Damping: Allows you to define the resulting damping of the part once the force has been applied to this part.

Instructor Notes:

Defining a Frequency Modulation

The modulation is absolutely needed to define the load excitation.

Reminder: You want to compute a harmonic dynamic case. It means, the modulation must describe the behavior of the amplitude versus the frequency. Thus, you will have create an Excel or txt file.

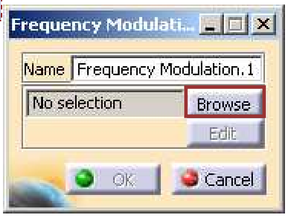
	A	B
1	X(Hz)	Y
2	0	0
3	320	0
4	320	1
5	450	1
6	450	0

Procedure to follow:

The first column must contain the frequency values. " (Hz)" must be written explicitly in the first cell. The values you define correspond to the excitation frequency range. You can put as many values as you want (the steps). Values must be in growing order.

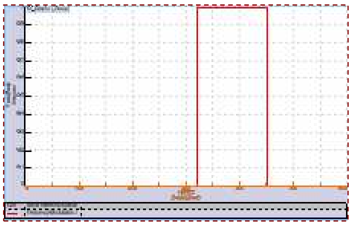
The second column contains the amplitudes of the excitation for each frequency.

It can take the values you want (must be positive).



Once the modulation file is ready, you have to import it.

You can check its content by click on edit.



Instructor Notes:

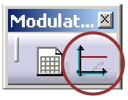
How to define White Noise Modulation

Remember that you have to import "modulation file" or define a "white noise modulation" to be able to define a "Load Excitation Case".

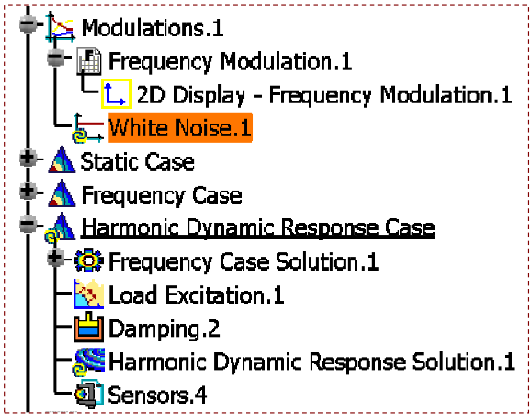
A White Noise Modulation is a particular case of frequency modulation:

The modulation is constant and equal to 1by default, used to get a uniform modulation.

1 Click on



2 Click on OK



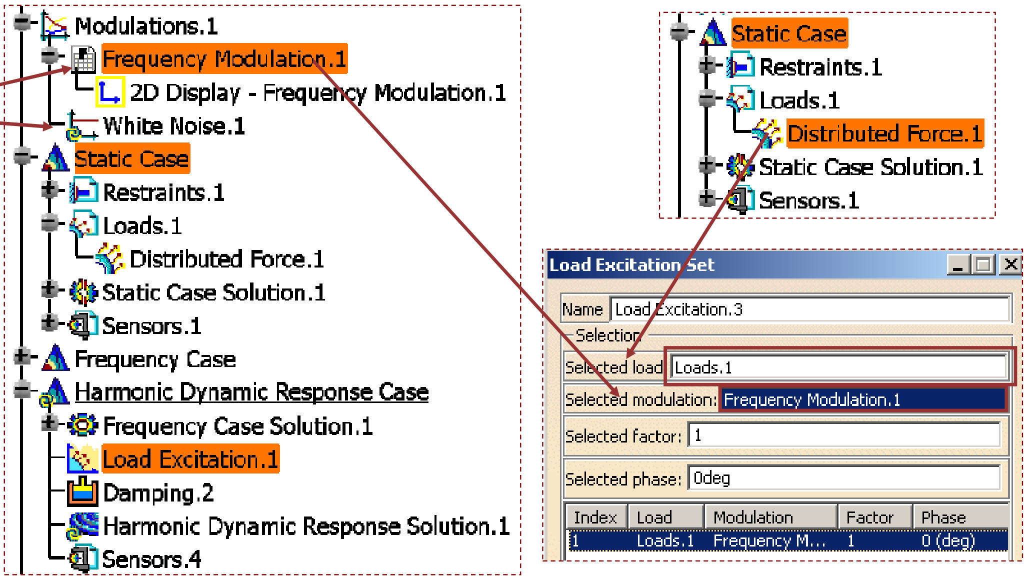
Instructor Notes:

Defining a Load Excitation Set Case

You can not define a Load Excitation if you have not previously defined a modulation.

Frequency Modulation :

- **From a file**
- **white Noise**



You must select the load on the previously computed static case. You must select a modulation: either a customized one (I.e from excel) or a white noise.

"Factor": Value by which you multiply the modulation.

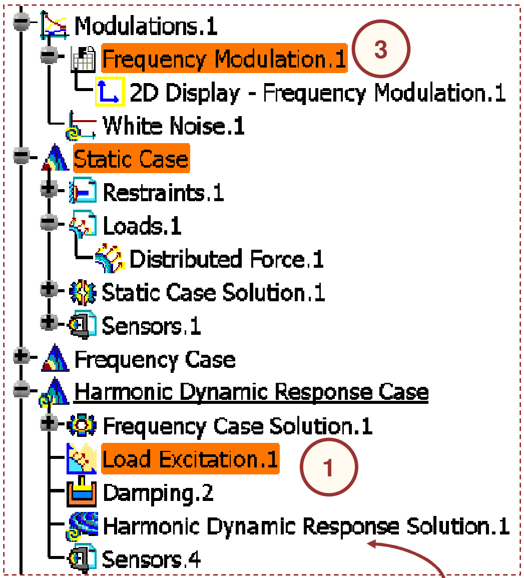
"Phase": you can associate a phase component of dynamic load excitation.

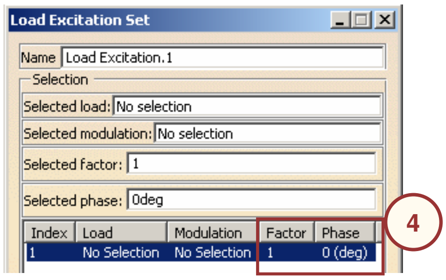
Instructor Notes:

How to Define a Load Excitation Set Case

Before you begin, you must have defined either a "White Noise modulation" (constant modulation) or have imported a modulation from a file. For more information, see the following job aids:

1 Double-click the load excitation set from the specification tree

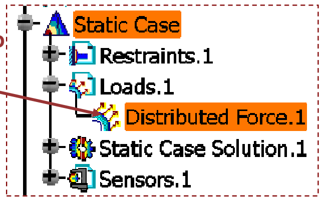




"Factor": Value by which you multiply the modulation

"Phase": you can associate a phase component of a dynamic load excitation

2 Select the load you want to excite in the specification tree



3 Select the desired modulation in the specification tree

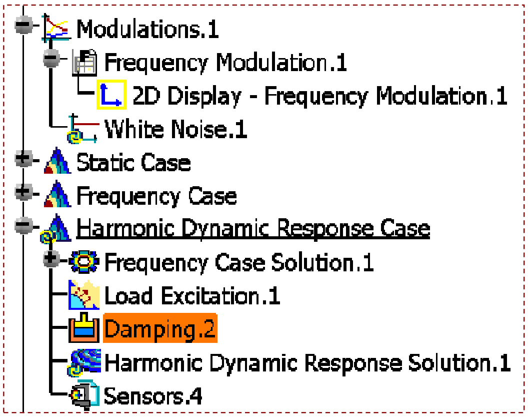
4 Define the desired Selected Factor

Instructor Notes:

How to Define a Damping Set (1/3)

Defining a Damping Set allows you to define the resulting damping of the part once the force has been applied to this part. You can choose between two damping types: Modal (default) or Rayleigh

It is absolutely necessary to define a damping set for dynamic analysis. However, in opposition to the modulation, a default value is automatically used if you do not define another one.



1 Double-click the damping set from the specification tree

The "damping" dialog box appears

2 Choose a damping type: "Modal" or "Rayleigh"

Instructor Notes:

How to Define a Damping Set (2/3)

- **Modal Damping**

The Modal Damping is a fraction of the critical damping. The critical damping is computed as follows:

$$Cr = 2\sqrt{mk}$$

where m is the mass of the system and k the stiffness of the system

3 Compute the frequency solution

(You have to compute the frequency solution before defining the damping parameters)

4 Click on the Component edition button to define the damping parameters

The Damping Definition dialog box appears

- **Rayleigh Damping**

The Rayleigh Damping is defined as follows:

$$[C] = \alpha[M] + \beta[K]$$

where [M] is the mass matrix, [K] is the stiffness matrix

3 Compute the frequency solution

(You have to compute the frequency solution before defining the damping parameters)

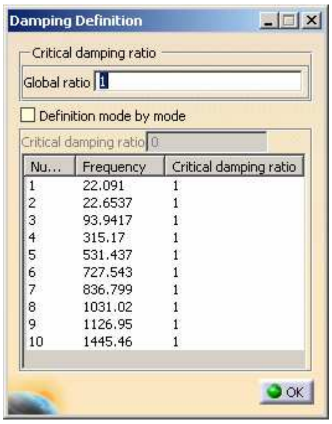
4 Click on the Component edition button to define the damping parameters

The Damping Definition dialog box appears

Instructor Notes:

How to Define a Damping Set (3/3)

Modal Damping



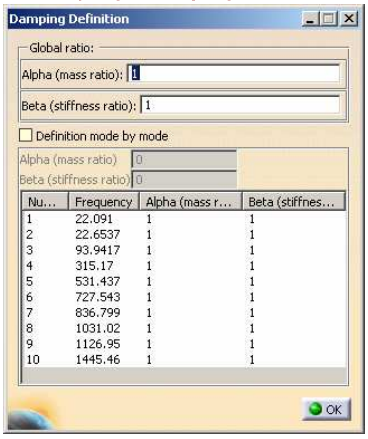
Global ratio: lets you define the factor of the critical damping for all the modes

Definition mode by mode: lets you define the critical damping ratio independently for each mode. Multi-selection is available in this case

5 Modify the modal damping parameters and click on OK

6 Click on OK in the Damping dialog box

Rayleigh Damping



Global ratio: lets you define the Alpha (mass ratio) and/or Beta (stiffness ratio) coefficients for all the modes

Definition mode by mode: lets you define the Alpha (mass ratio) and/or Beta (stiffness ratio) coefficients independently for each selected mode. Multi-selection is available in this Case

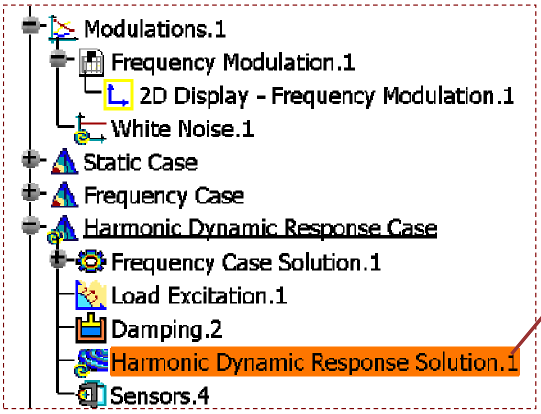
5 Modify the Rayleigh damping parameters and click on OK

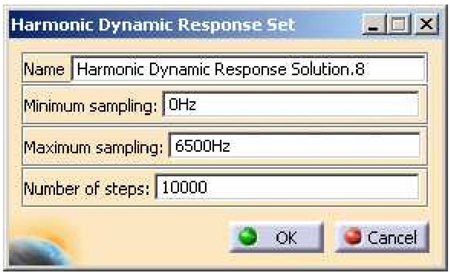
6 Click OK in the Damping dialog box

Instructor Notes:

Harmonic Dynamic Response Set

Before you compute an analysis, you have to specify the frequency sampling.





The response is computed in the frequency domain on a regular sampling.

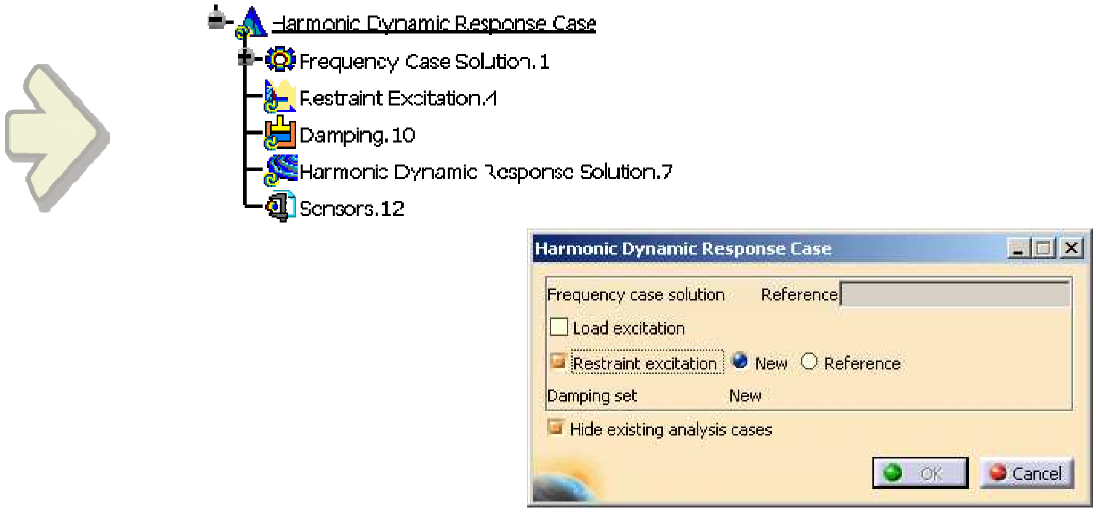
The minimum and the maximum sampling correspond to the lower and the upper bounds of the frequency range of interest. The Number of steps corresponds to the number of calculated points inside the frequency range.

To get more precision in the peak value, you can either increase the number of steps inside the frequency range, or focus on the peak by choosing a smaller frequency range. The second solution is better.

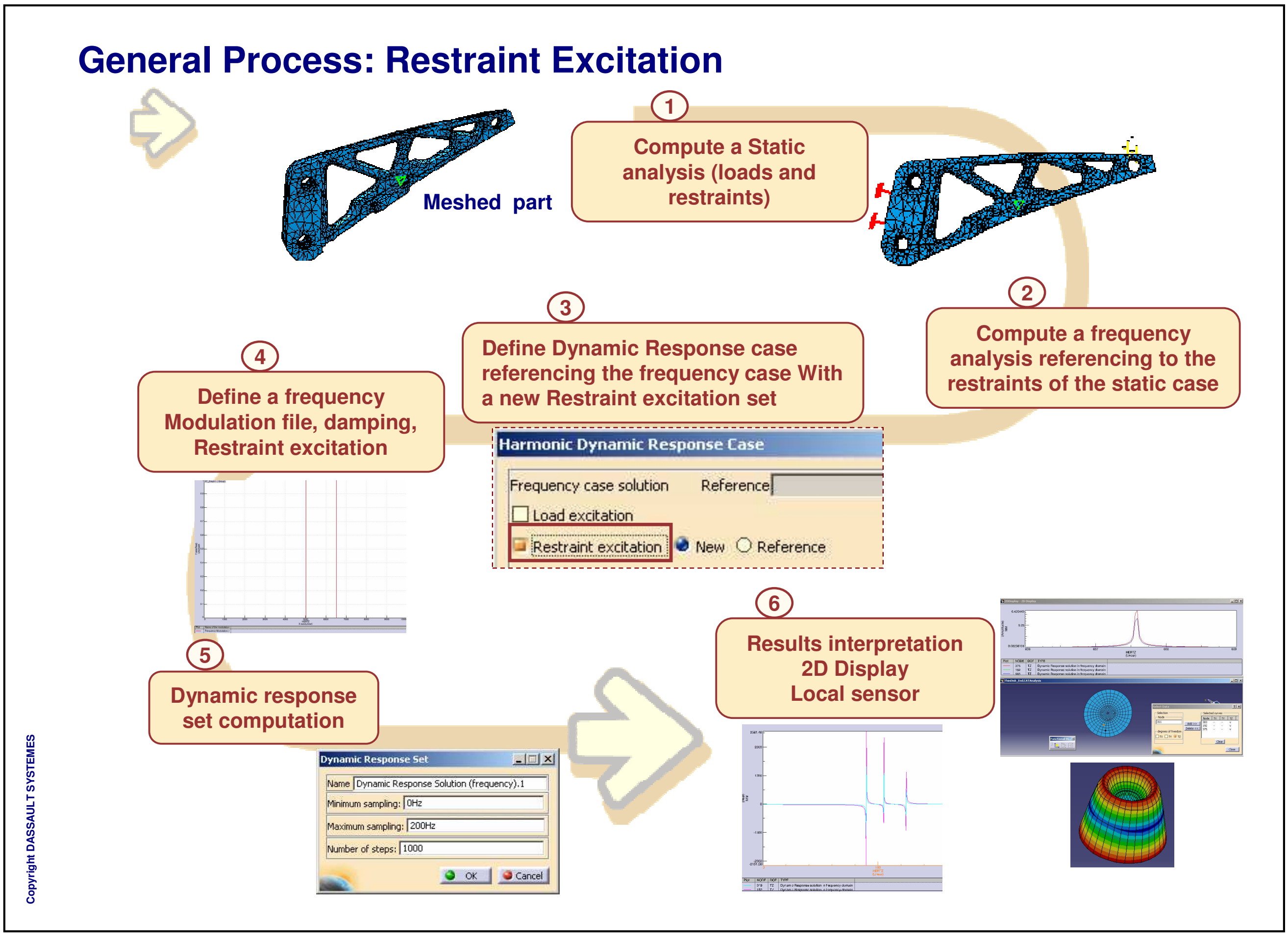
Instructor Notes:

Harmonic: Restraint Excitation Case

You will see how to define a Harmonic Response Case with a Restraint Excitation



Instructor Notes:



Instructor Notes:

- Donner des procès généraux quand cela est possible pour illustrer
- What , How, Why ne sont pas obligatoires mais doivent guider et aider dans la rédaction du foils.

Forme

- ATENTION de na pas modifier la taille du cadre intérieur de la diapo qui est dimensionne pour pouvoir imprimer en format Américain aussi bien que Européen
- Essayer de respecter la palette de couleurs proposée
- Ne pas oublier de mettre a jour le titre du cours dans le masque
- Chaque mot du titre doit commencer par une majuscule

Style:

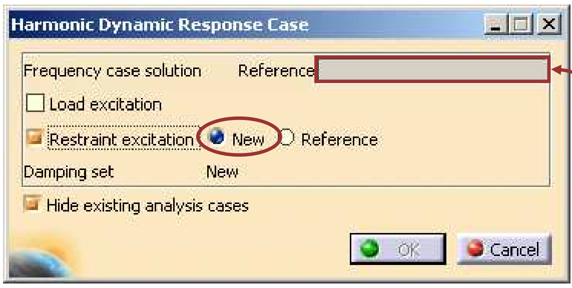
Utiliser You... A la place des phrases impersonnelles

Utiliser la voix active plutot que passive

Ecrire des phrases simples: Souvenez vous que ce document peut etre traduit en d'autres langues et donc nous devons eviter toutes ambiguites.

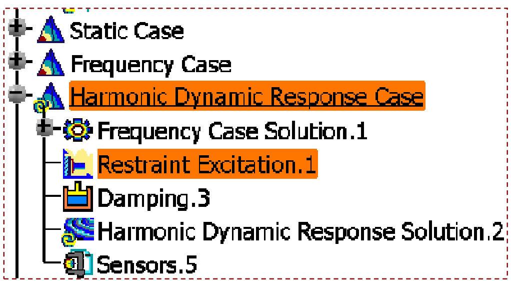
Defining a Harmonic/Restraint Excitation Case

When defining a Dynamic Response Case, you have to select different data.



You must refer to the pre-computed frequency analysis to take into account the modal frequencies of the system.

When you define a dynamic case for the first time, you have to check "New" restraint excitation.



On the left, you can see a typical tree of harmonic/Restraint Excitation Case. Below are the main components:

First, you can see the frequency Case you have referred and the different solutions you had displayed.

Restraint Excitation: that is a very important thing. It will refer to the restraint defined in the static analysis and that is on it that you will apply a modulation and an acceleration. It means, you will have to define a modulation file.

Damping: Allows you to define the resulting damping of the part once the force has been applied to this part.

You need to compute a static and a frequency analysis.

Instructor Notes:

Defining a Frequency Modulation

The modulation is absolutely needed to define a restraint excitation.

Reminder: You want to compute a harmonic dynamic case. It means, the modulation must describe the behavior of the amplitude versus the frequency. Thus, you will have create an Excel or txt file.

	A	B
1	X(Hz)	Y
2	0	0
3	320	0
4	320	1
5	450	1
6	450	0

Procedure to follow:

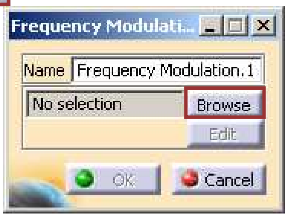
The first column must contain the frequency values. " (Hz)" must be written explicitly in the first cell. The values you define correspond to the excitation frequency range. You can put as many values as you want (the steps). Values must be in growing order.

The second column contains the amplitudes of the excitation for each frequency.

It can take the values you want (must be positive).

Once the modulation file is ready, you have to import it.

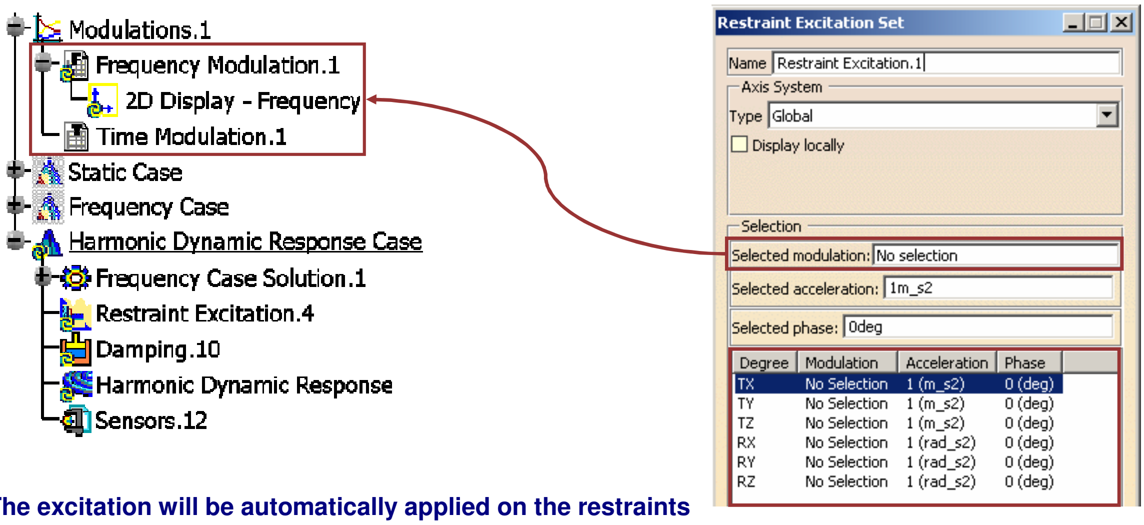
You can check its content by click on edit.



Instructor Notes:

Defining a Restraint Excitation Set Case

You can not define a Restraint Excitation if you have not previously defined a modulation



The excitation will be automatically applied on the restraints of the static case. You must select a modulation, but you can not use a white noise. You will have to apply a modulation and specify an acceleration for the rigid-body motions of the support (RX, RY, RZ, TX, TY, TZ) of your choice.

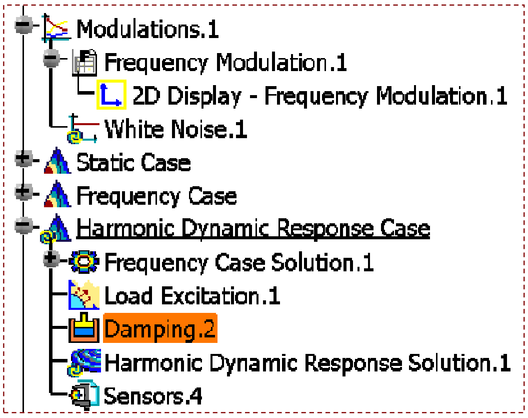
"Phase": you can associate a phase with each rigid-body.

Instructor Notes:

How to Define a Damping Set (1/3)

Defining a Damping Set allows you to define the resulting damping of the part once the force has been applied to this part. You can choose between two damping types: Modal (default) or Rayleigh

It is absolutely necessary to define a damping set for dynamic analysis. However, in opposition to the modulation, a default value is automatically used if you do not define another one.



1 **Double-click the damping set from the specification tree**

The "damping" dialog box appears

2 **Choose a damping type: "Modal" or "Rayleigh"**

Instructor Notes:

How to Define a Damping Set (2/3)

- **Modal Damping**

The Modal Damping is a fraction of the critical damping. The critical damping is computed as follows:

$$Cr = 2\sqrt{mk}$$

where m is the mass of the system and k the stiffness of the system

3 **Compute the frequency solution**

(You have to compute the frequency solution before defining the damping parameters)

4 **Click on the Component edition button to define the damping parameters**

The Damping Definition dialog box appears

- **Rayleigh Damping**

The Rayleigh Damping is defined as follows:

$$[C] = \alpha[M] + \beta[K]$$

where [M] is the mass matrix, [K] is the stiffness matrix

3 **Compute the frequency solution**

(You have to compute the frequency solution before defining the damping parameters)

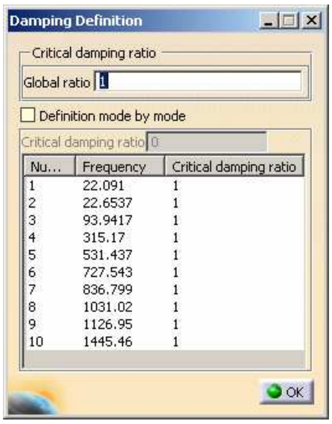
4 **Click on the Component edition button to define the damping parameters**

The Damping Definition dialog box appears

Instructor Notes:

How to Define a Damping Set (3/3)

- **Modal Damping**



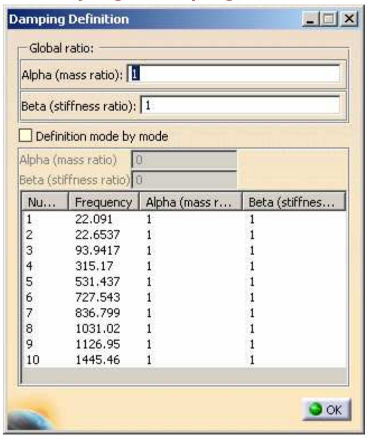
Global ratio: lets you define the factor of the critical damping for all the modes

Definition mode by mode: lets you define the critical damping ratio independently for each mode. Multi-selection is available in this case

5 **Modify the modal damping parameters and click on OK**

6 **Click on OK in the Damping dialog box**

- **Rayleigh Damping**



Global ratio: lets you define the Alpha (mass ratio) and/or Beta (stiffness ratio) coefficients for all the modes

Definition mode by mode: lets you define the Alpha (mass ratio) and/or Beta (stiffness ratio) coefficients independently for each selected mode. Multi-selection is available in this Case

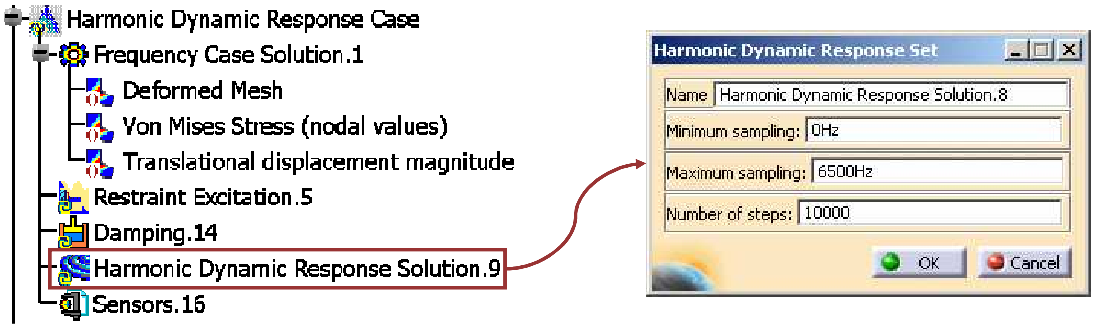
5 **Modify the Rayleigh damping parameters and click on OK**

6 **Click OK in the Damping dialog box**

Instructor Notes:

Harmonic Dynamic Response Set

Before you compute an analysis, you have to specify the frequency sampling.



The response is computed in the frequency domain on a regular sampling.

The minimum and the maximum sampling correspond to the lower and the upper bounds of the frequency range of interest. The Number of steps corresponds to the number of calculated points inside the frequency range.

To get more precision in the peak value, you can either increase the number of steps inside the frequency range, or focus on the peak by choosing a smaller frequency range. The second solution is better.

Instructor Notes:

To Sum Up ...

In the lesson, you have seen how to define 2 types of Harmonic Dynamic Analysis :

- **Load Excitation Case**
- **Restraint Excitation Case**

Instructor Notes:

Transient Response Case

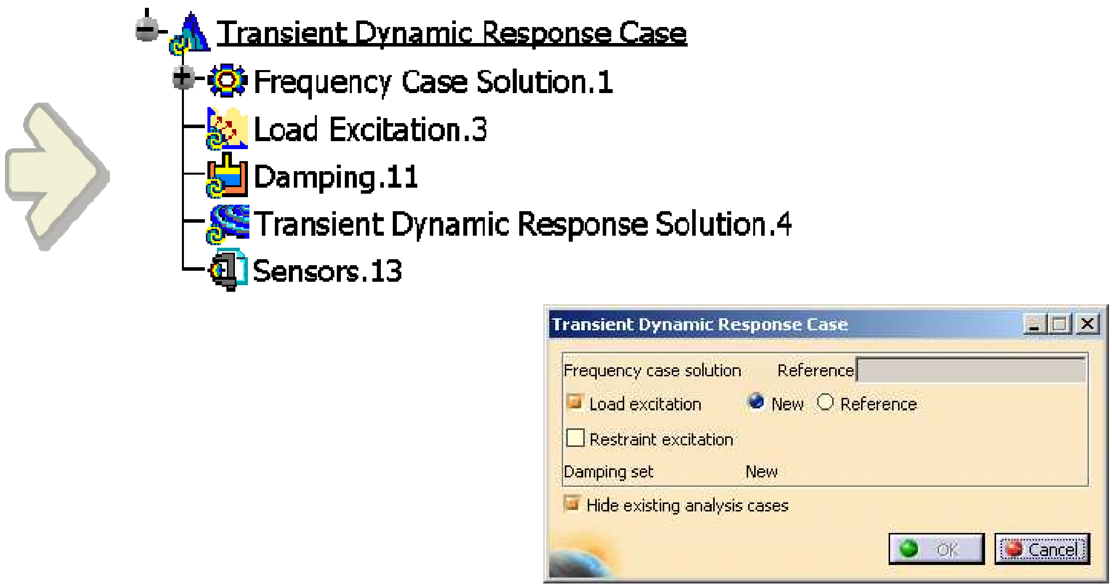
You will see different excitation cases used in Transient Response Analysis

- Transient: Load Excitation Case
- Transient: Restraint Excitation Case
- To Sum Up

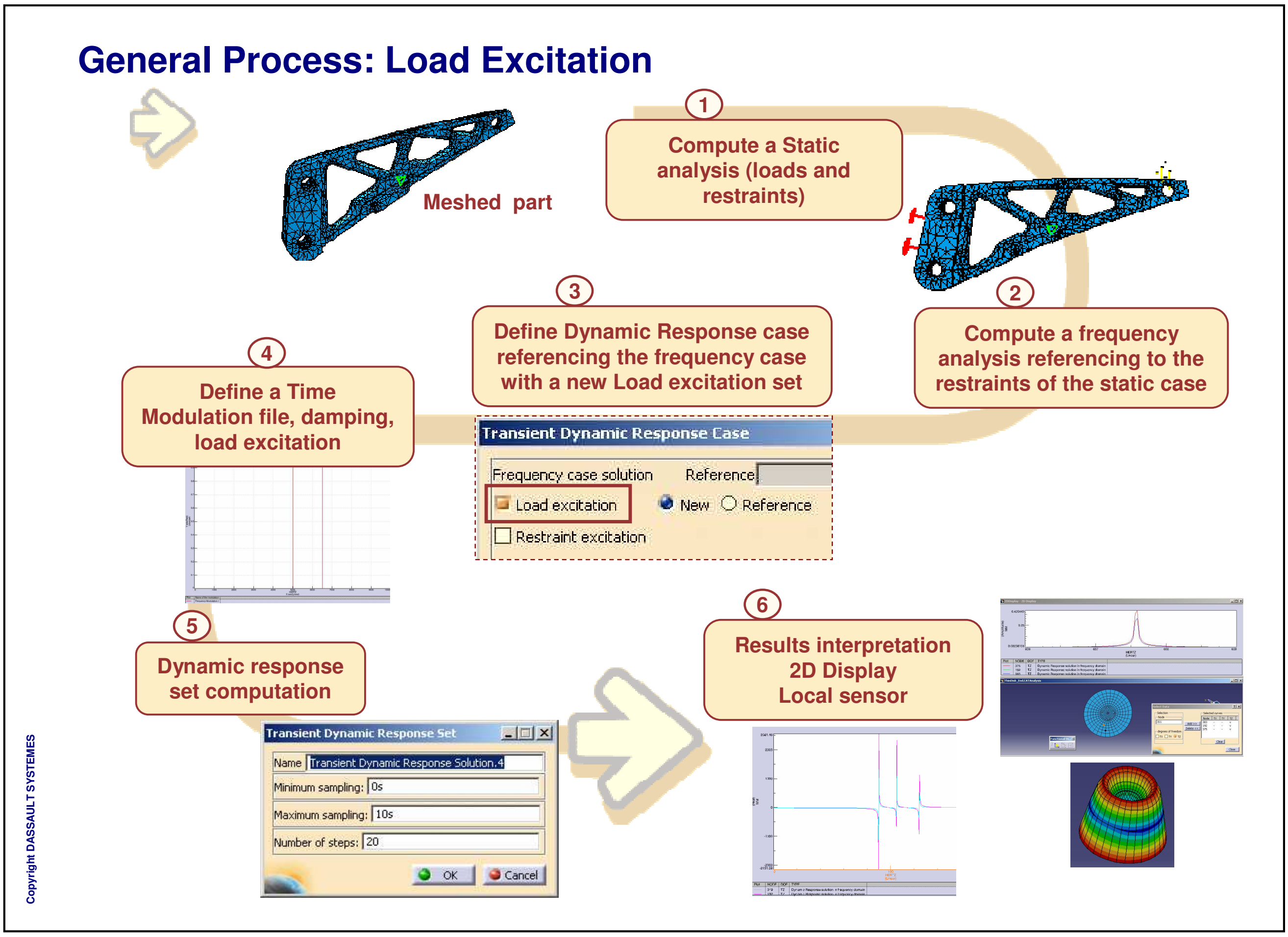
Instructor Notes:

Transient: Load Excitation Case

You will see how to define a Transient Response Case with a Load Excitation



Instructor Notes:



Instructor Notes:

- Donner des procès généraux quand cela est possible pour illustrer
- What , How, Why ne sont pas obligatoires mais doivent guider et aider dans la rédaction du foils.

Forme

- ATENTION de na pas modifier la taille du cadre intérieur de la diapo qui est dimensionne pour pouvoir imprimer en format Américain aussi bien que Européen
- Essayer de respecter la palette de couleurs proposée
- Ne pas oublier de mettre a jour le titre du cours dans le masque
- Chaque mot du titre doit commencer par une majuscule

Style:

Utiliser You... A la place des phrases impersonnelles

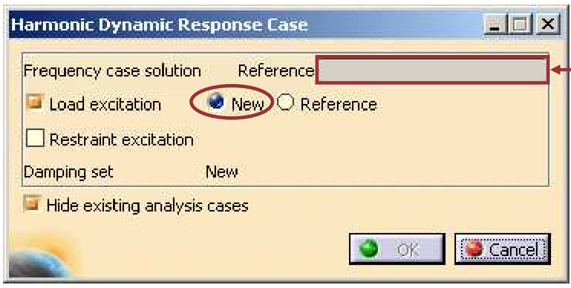
Utiliser la voix active plutot que passive

Ecrire des phrases simples: Souvenez vous que ce document peut etre traduit en d'autres langues et donc nous devons eviter toutes ambiguites.

Defining a Transient/Load Excitation Case

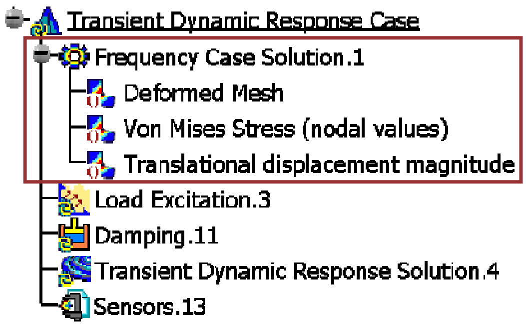
When defining a "Transient Dynamic Response Case" you have to select different data.

Reminder: You must have compute a static and a frequency analysis.



You must refer to the pre-computed frequency analysis to take into account the modal frequencies of the system.

When you define a dynamic case for the first time, you have to check "New" load excitation.



On the left, you can see a typical tree of transient/load Excitation Case. Below are the main components:

First, you can see the frequency Case you have referred it and the different solutions you had displayed.

Load Excitation: that is a very important thing. It will refer to the load of the pre-computed static analysis and that is on this load that you will apply a time modulation. It means, you will have to define a time modulation file. (cf. jobaids presented further)

Damping: Allows you to define the resulting damping of the part once the force has been applied to this part.

Instructor Notes:

Defining a Time Modulation

In a transient dynamic case, modulation describes the behavior of the amplitude versus TIME. Thus, you will have create an Excel or txt file. The modulation is absolutely needed to define the load excitation.

	A	B
1	X(s)	Signal
2	0	0
3	5	1
4	10	1
5	15	2
6	15	1
7	20	1
8	20	0

Procedure to follow:

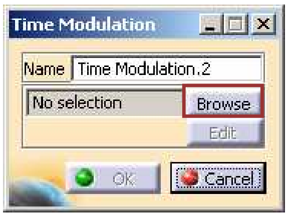
The first column must contain the time values. “(s)” must be written explicitly in the first cell. You can put as many values as you want (the steps).

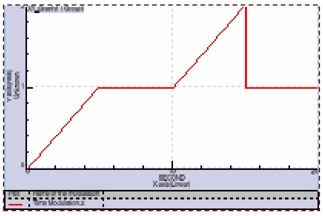
The second column contains the amplitudes of the excitation for each time step.

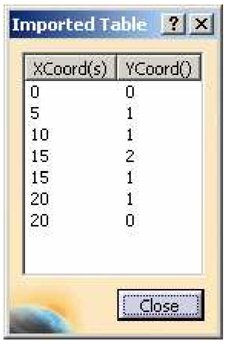
It can take the values you want (must be positive).

Once the time modulation file is ready, you have to import it.

You can check its content by click on edit.



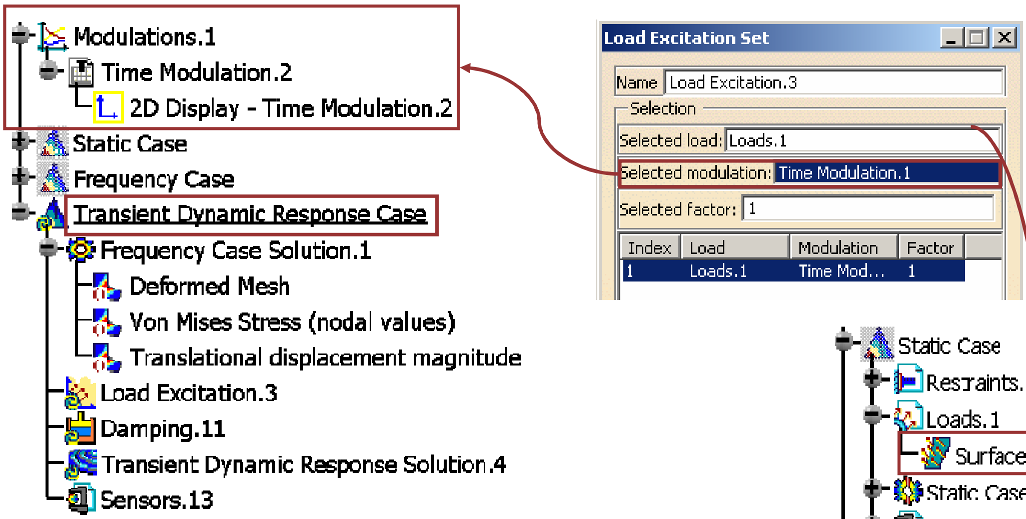


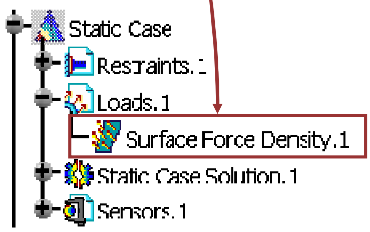


Instructor Notes:

Defining a Load Excitation Set Case

You can not define a Load Excitation if you have not previously defined a modulation.





You must select the load on the previously computed static case.

You must select a TIME modulation (from a file).

The selected modulation will apply on the selected load.

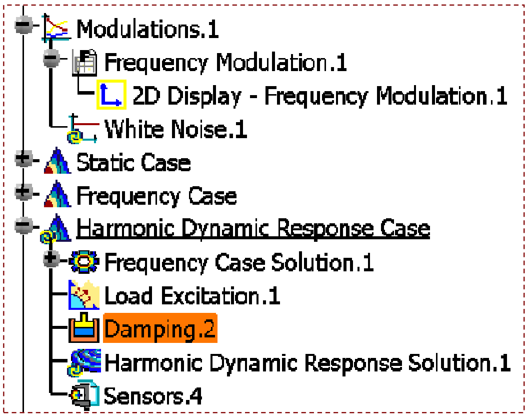
“Factor”: Value by which you multiply the modulation.

Instructor Notes:

How to Define a Damping Set (1/3)

Defining a Damping Set allows you to define the resulting damping of the part once the force has been applied to this part. You can choose between two damping types: Modal (default) or Rayleigh

It is absolutely necessary to define a damping set for dynamic analysis. However, in opposition to the modulation, a default value is automatically used if you do not define another one.



1 **Double-click the damping set from the specification tree**

The "damping" dialog box appears

2 **Choose a damping type: "Modal" or "Rayleigh"**

Instructor Notes:

How to Define a Damping Set (2/3)

- **Modal Damping**

The Modal Damping is a fraction of the critical damping. The critical damping is computed as follows:

$$Cr = 2\sqrt{mk}$$

where m is the mass of the system and k the stiffness of the system

3 Compute the frequency solution

(You have to compute the frequency solution before defining the damping parameters)

4 Click on the Component edition button to define the damping parameters

The Damping Definition dialog box appears

- **Rayleigh Damping**

The Rayleigh Damping is defined as follows:

$$[C] = \alpha[M] + \beta[K]$$

where [M] is the mass matrix, [K] is the stiffness matrix

3 Compute the frequency solution

(You have to compute the frequency solution before defining the damping parameters)

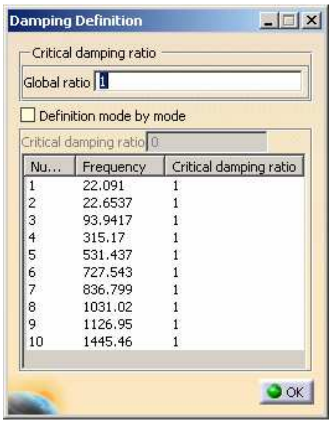
4 Click on the Component edition button to define the damping parameters

The Damping Definition dialog box appears

Instructor Notes:

How to Define a Damping Set (3/3)

Modal Damping



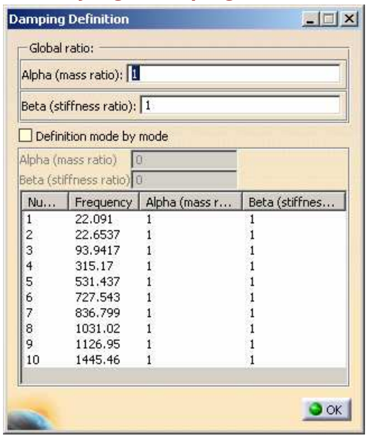
Global ratio: lets you define the factor of the critical damping for all the modes

Definition mode by mode: lets you define the critical damping ratio independently for each mode. Multi-selection is available in this case

5 Modify the modal damping parameters and click on OK

6 Click on OK in the Damping dialog box

Rayleigh Damping



Global ratio: lets you define the Alpha (mass ratio) and/or Beta (stiffness ratio) coefficients for all the modes

Definition mode by mode: lets you define the Alpha (mass ratio) and/or Beta (stiffness ratio) coefficients independently for each selected mode. Multi-selection is available in this Case

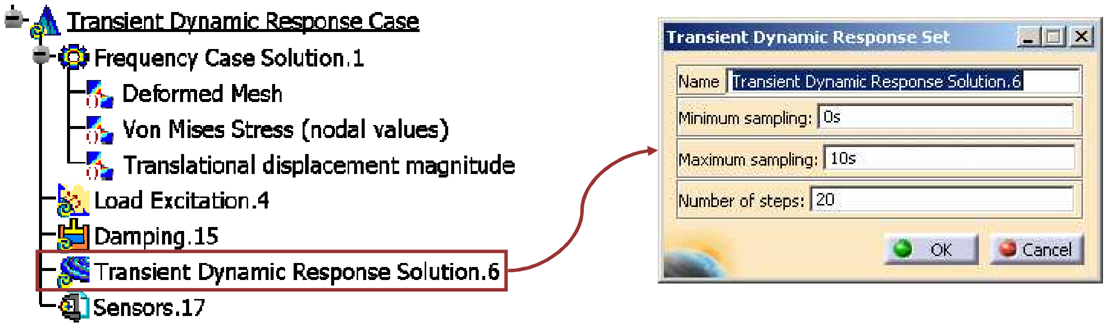
5 Modify the Rayleigh damping parameters and click on OK

6 Click OK in the Damping dialog box

Instructor Notes:

Transient Dynamic Response Set

Before you compute an analysis, you have to specify the time sampling.



The response is computed in the time domain on a regular sampling.

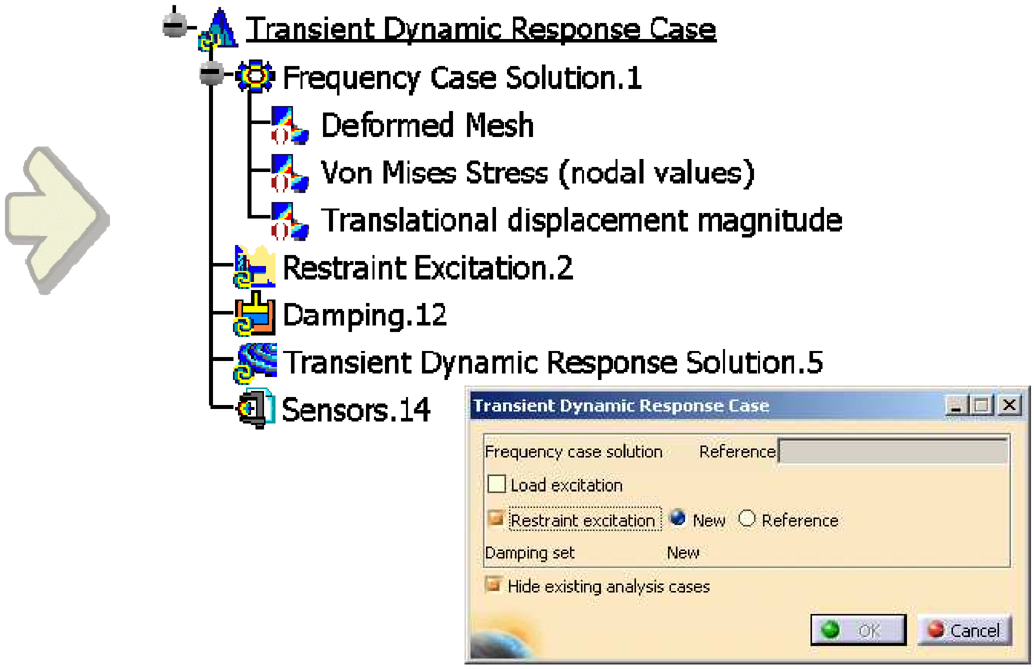
The minimum and the maximum sampling correspond to the lower and the upper bounds of the time range of interest. The Number of steps corresponds to the number of calculated points inside the time range.

To get more precision in the peak value, you can either increase the number of steps inside the time range, or focus on the peak by choosing a smaller time range. The second solution is better.

Instructor Notes:

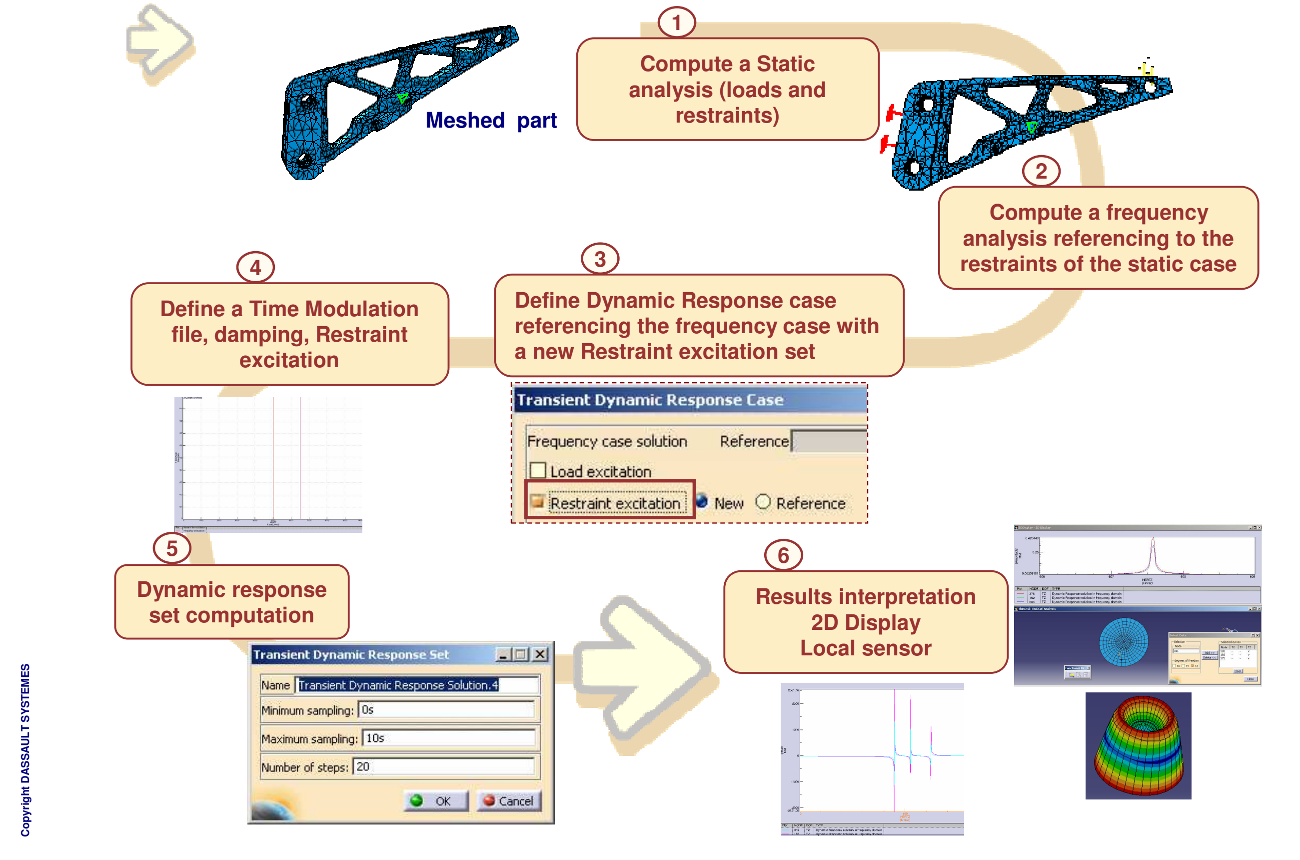
Transient: Restraint Excitation Case

You will see how to define a Transient Response Case with a Restraint Excitation



Instructor Notes:

General Process: Restraint Excitation



Instructor Notes:

- Donner des procès généraux quand cela est possible pour illustrer
- What , How, Why ne sont pas obligatoires mais doivent guider et aider dans la rédaction du foils.

Forme

- ATENTION de na pas modifier la taille du cadre intérieur de la diapo qui est dimensionne pour pouvoir imprimer en format Américain aussi bien que Européen
- Essayer de respecter la palette de couleurs proposée
- Ne pas oublier de mettre a jour le titre du cours dans le masque
- Chaque mot du titre doit commencer par une majuscule

Style:

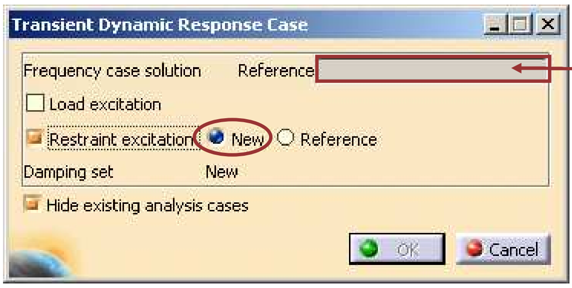
Utiliser You... A la place des phrases impersonnelles

Utiliser la voix active plutot que passive

Ecrire des phrases simples: Souvenez vous que ce document peut etre traduit en d'autres langues et donc nous devons eviter toutes ambiguites.

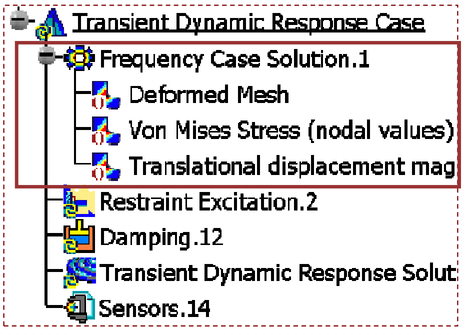
Defining a Transient/Restraint Excitation Case

When defining a"Dynamic Response Case" you have to select different data.



You must refer to the pre-computed frequency analysis to take into account the modal frequencies of the system.

When you define a dynamic case for the first time, you have to check "New" restraint excitation.



On the left, you can see a typical tree of Transient/Restraint Excitation Case. Below are the main components:

First, you can see the frequency Case you have referred and below, the different solutions you had displayed.

Restraint Excitation: that is a very important thing. It will refer to the restraint of the pre-computed static analysis and it is on this load that you will apply a time modulation and an acceleration.

Damping: Allows to define the resulting damping of the part once the force has been applied to this part.

You need to compute a static and a frequency analysis.

Instructor Notes:

Defining a Time Modulation

In a transient dynamic case, Time Modulation describes the behavior of the amplitude versus TIME. Thus, you will have create an Excel or txt file. The modulation is absolutely needed to define the load excitation.

	A	B
1	X(s)	Signal
2	0	0
3	5	1
4	10	1
5	15	2
6	15	1
7	20	1
8	20	0

Procedure to follow:

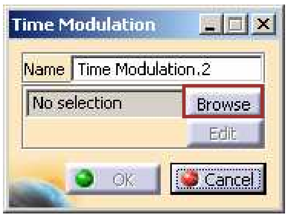
The first column must contain the time values. "(s)" must be written explicitly in the first cell. You can put as many values as you want (the steps).

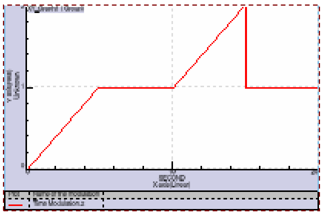
The second column contains the amplitudes of the excitation for each time step.

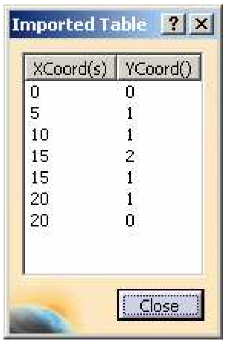
It can take the values you want (must be positive).

Once the time modulation file is ready, you have to import it.

You can check its content by click on edit.



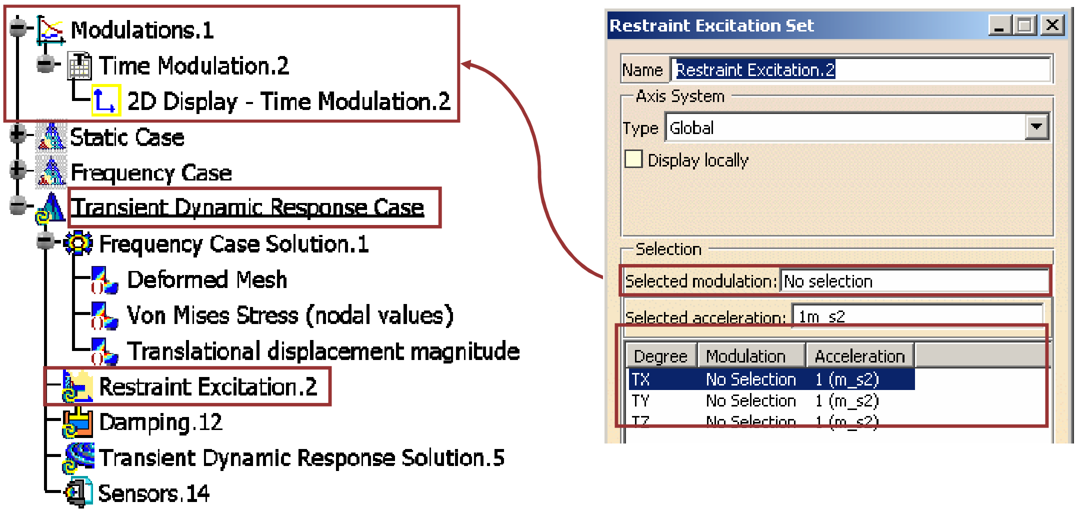




Instructor Notes:

Defining a Restraint Excitation Set Case

You can not define a Restraint Excitation if you have not previously defined a time modulation.



The excitation will be automatically applied on the restraints of the static case. You must select a TIME modulation.

You will have to apply this time modulation on the rigid-body motions of the support (TX, TY, TZ) of your choice and, specify an acceleration.

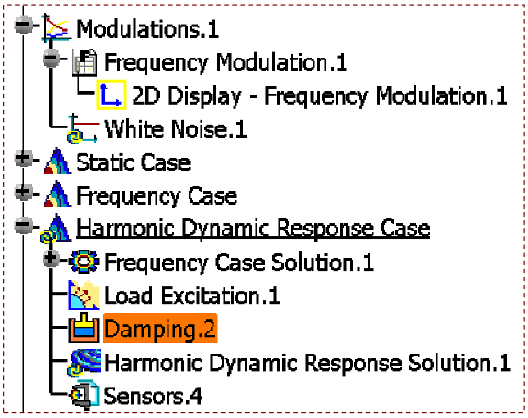
"Phase": you can associate a phase with each rigid-body

Instructor Notes:

How to Define a Damping Set (1/3)

Defining a Damping Set allows you to define the resulting damping of the part once the force has been applied to this part. You can choose between two damping types: Modal (default) or Rayleigh

It is absolutely necessary to define a damping set for dynamic analysis. However, in opposition to the modulation, a default value is automatically used if you do not define another one.



1 **Double-click the damping set from the specification tree**

The "damping" dialog box appears

2 **Choose a damping type: "Modal" or "Rayleigh"**

Instructor Notes:

How to Define a Damping Set (2/3)

- **Modal Damping**

The Modal Damping is a fraction of the critical damping. The critical damping is computed as follows:

$$Cr = 2\sqrt{mk}$$

where m is the mass of the system and k the stiffness of the system

3 Compute the frequency solution

(You have to compute the frequency solution before defining the damping parameters)

4 Click on the Component edition button to define the damping parameters

The Damping Definition dialog box appears

- **Rayleigh Damping**

The Rayleigh Damping is defined as follows:

$$[C] = \alpha[M] + \beta[K]$$

where [M] is the mass matrix, [K] is the stiffness matrix

3 Compute the frequency solution

(You have to compute the frequency solution before defining the damping parameters)

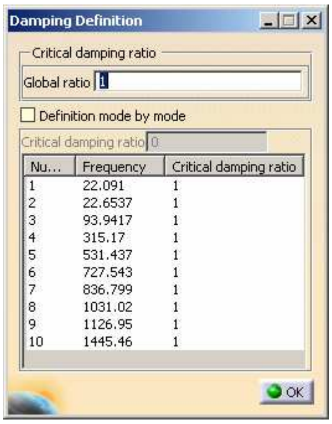
4 Click on the Component edition button to define the damping parameters

The Damping Definition dialog box appears

Instructor Notes:

How to Define a Damping Set (3/3)

- **Modal Damping**



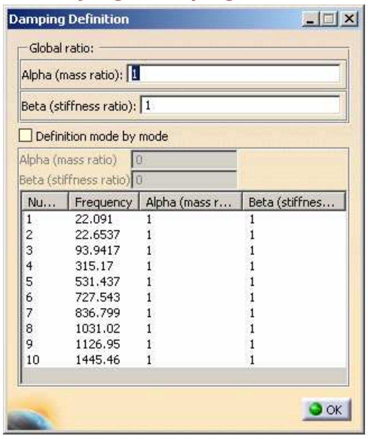
Global ratio: lets you define the factor of the critical damping for all the modes

Definition mode by mode: lets you define the critical damping ratio independently for each mode. Multi-selection is available in this case

(5) Modify the modal damping parameters and click on OK

(6) Click on OK in the Damping dialog box

- **Rayleigh Damping**



Global ratio: lets you define the Alpha (mass ratio) and/or Beta (stiffness ratio) coefficients for all the modes

Definition mode by mode: lets you define the Alpha (mass ratio) and/or Beta (stiffness ratio) coefficients independently for each selected mode. Multi-selection is available in this Case

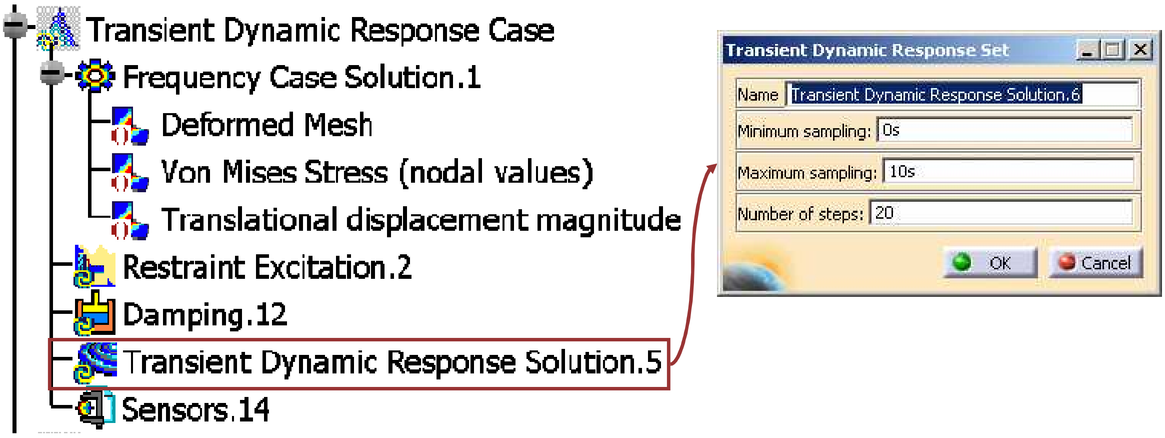
(5) Modify the Rayleigh damping parameters and click on OK

(6) Click OK in the Damping dialog box

Instructor Notes:

Transient Dynamic Response Set

Before you compute an analysis, you have to specify the time sampling.



The response is computed in the time domain on a regular sampling.

The minimum and the maximum sampling correspond to the lower and the upper bounds of the time range of interest. The Number of steps corresponds to the number of calculated points inside the time range.

To get more precision in the peak value, you can either increase the number of steps inside the time range, or focus on the peak by choosing a smaller time range. The second solution is better.

Instructor Notes:

To Sum Up ...

In the lesson, you have seen how to define two types of Transient Dynamic Analysis :

- **Load Excitation Case**
- **Restraint Excitation Case**

Instructor Notes:

To Sum Up ...

In the lesson, you have seen how to define two types of Dynamic Analysis :

- **Harmonic Case**
- **Transient Case**

Instructor Notes:

Results Visualization

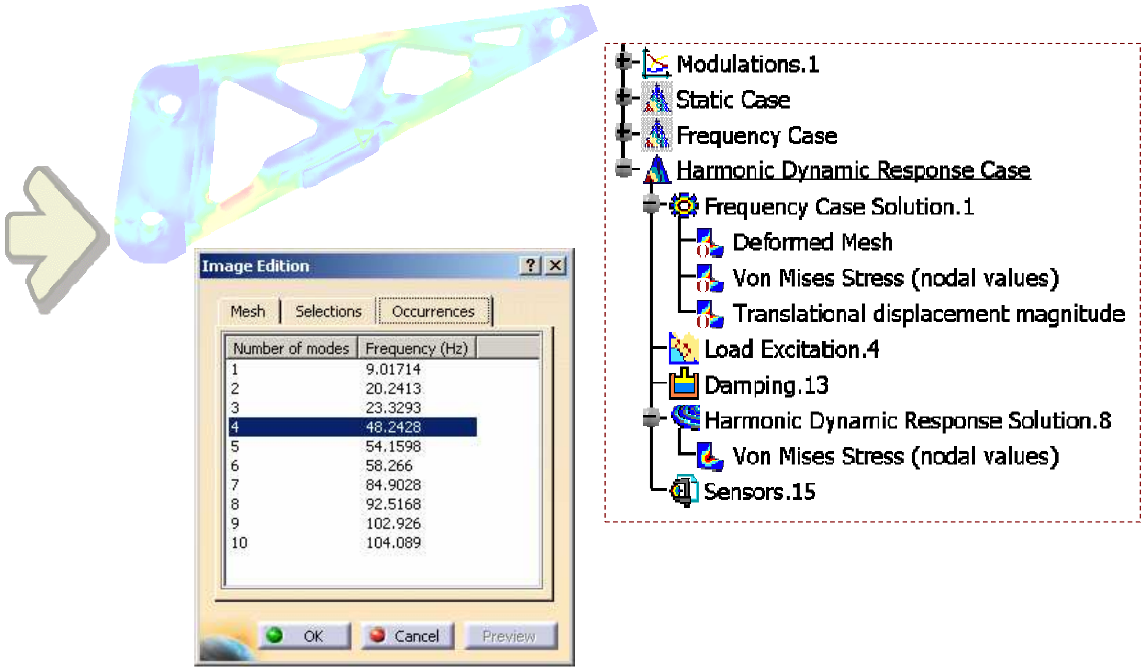
In this lesson, you will see how to visualize GDY results

- Generating Images
- Generating 2D Display
- To Sum Up

Instructor Notes:

Generating Images

You will see how to generate images corresponding to analysis results.



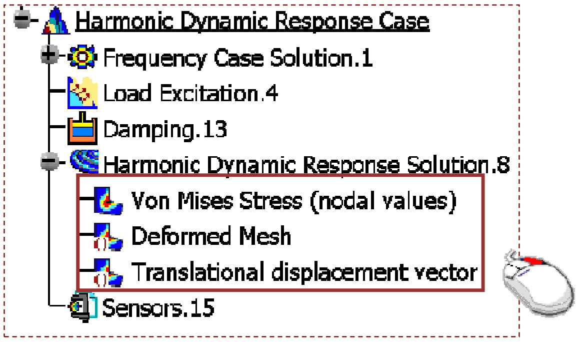
Instructor Notes:

Introduction

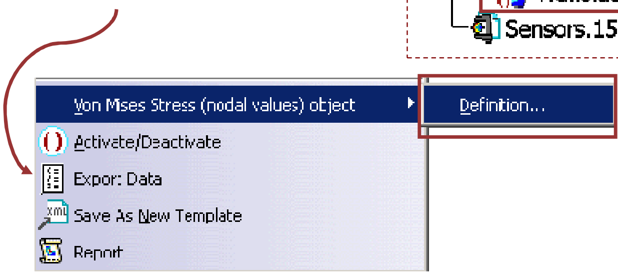
You will see the different way to visualize results. To display Dynamic Analysis results you will use the same tools as the ones you use under GPS/EST License.

Whatever the dynamic case you have computed, you can visualize:

- The deformed mesh
- The Von Mises Stress
- The Displacements
- The Principal Stress
- The Precision



As under GPS, the contextual menu is available and allows you to customize the visualization:

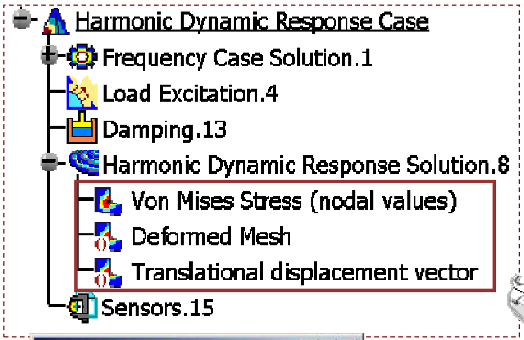


Instructor Notes:

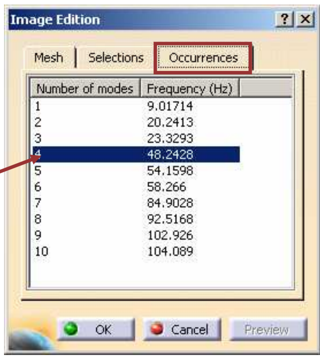
Harmonic Dynamic Analysis Results Visualization

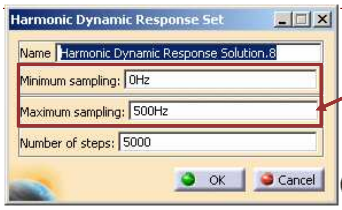
You will see how to display results according to the frequency.

It means, you don't want to display the results of the static case only, neither the results of the frequency case only but, a combination of both, according to your modulation file and the frequency sampling you have defined for the computation.



By double-clicking on the results (in the tree) or using the contextual menu, the image edition dialog box will allow you to choose the occurrences. Just select the one you want and corresponding images will be automatically displayed.





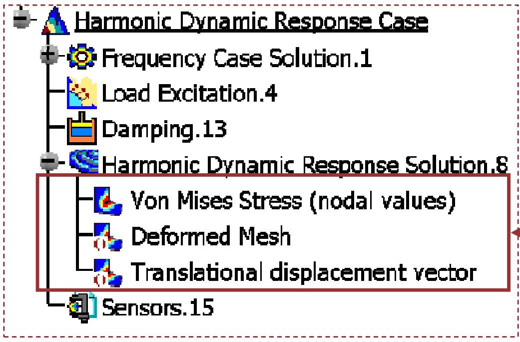
You can not visualize occurrences results that are out of the frequency range.

Harmonic dynamic analysis is a combination between a static case and a frequency case on which you have applied a frequency modulation.

Instructor Notes:

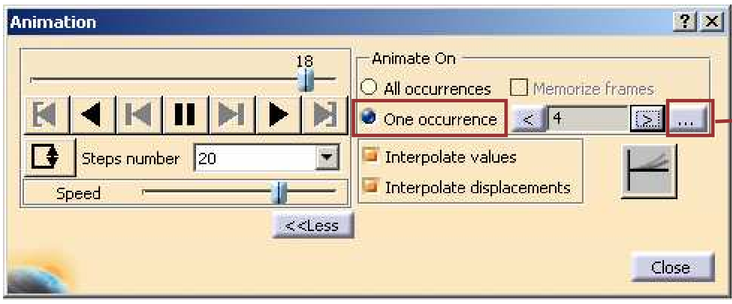
Animate Images

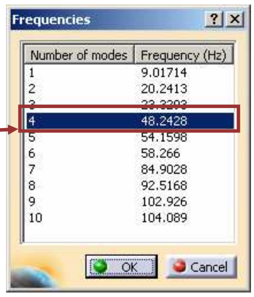
You will see how to animate images according to the occurrences of your choice. To display GDY results you will use the same tools as the ones you use under GPS/EST.



Display the image of your choice

You can select all the frequencies available or animate the occurrence of your choice

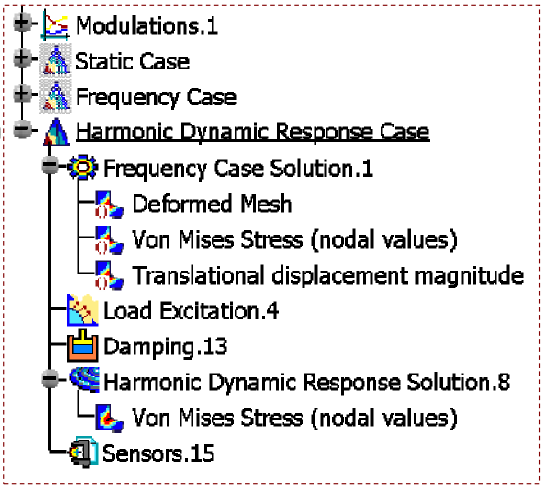


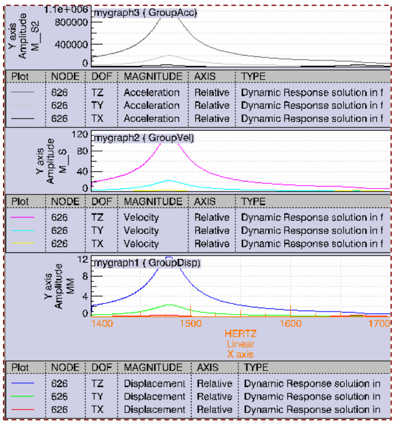


Instructor Notes:

Generating 2D Display

You will see how to generate 2D display for local results exploitations

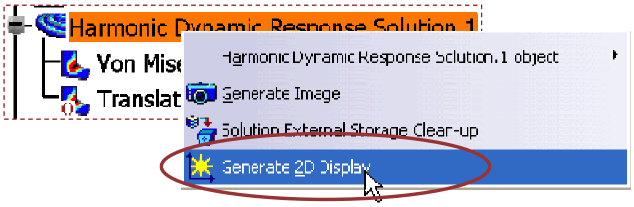




Instructor Notes:

What are 2D Displays

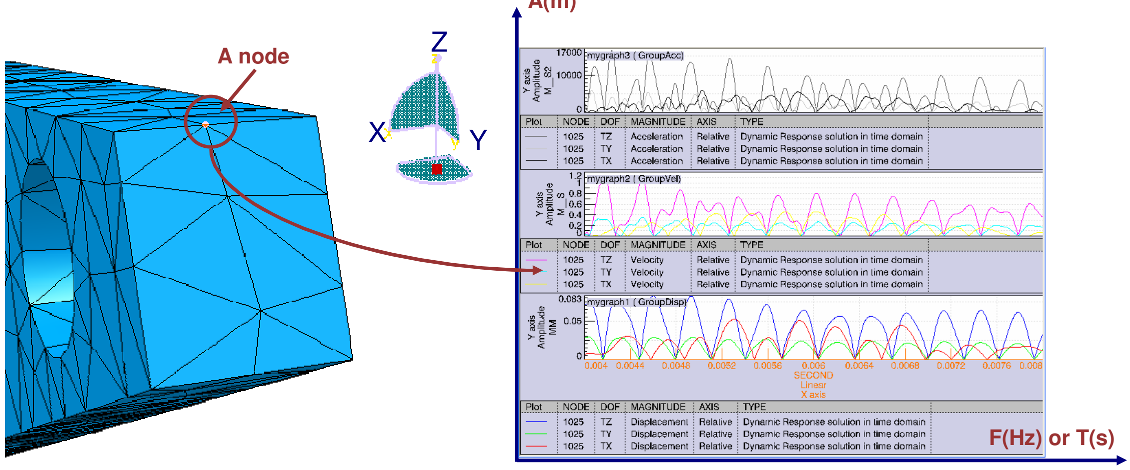
The "Generate 2D-Display" tool is available in the contextual menu of the dynamic response solution sets.



The "2D display" tool allows you to know :

- **Translations (X, Y, Z)**
- **Translational velocity**
- **Translation acceleration**

of any nodes of the mesh and also to visualize their behaviors versus time or frequency (according to the kind of dynamic case you have computed)



Instructor Notes:

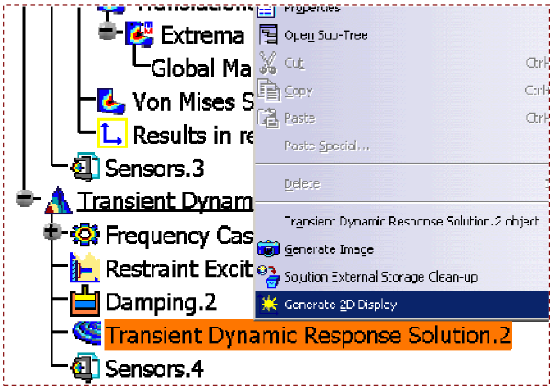
Two Types of Axis Systems

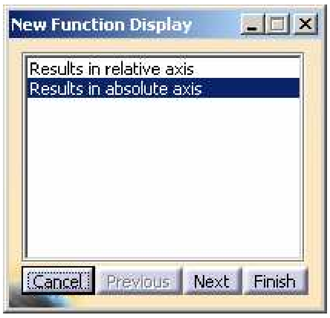
You can display 2D Curves results either in relative axis or in absolute axis.

When you define a dynamic analysis (Transient or Harmonic case), you have to choose between two types of excitation:

- **Load**
- **Restraint**

If you choose the restraint excitation, you will be able to visualize results (using 2D Display tool), either:





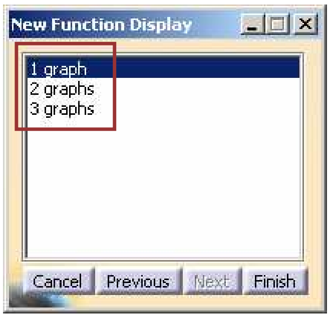
It will allows you to visualize the deformation without the rigid body motion. (it can be useful to display the flexible part of the deformation only)

Instructor Notes:

2D Output: Three Types of Results

For each node, you can now display translational velocity and translational acceleration.

When defining a 2D display, the dialog box below asks you to choose between:



Meaning:

1 Graph: Translations

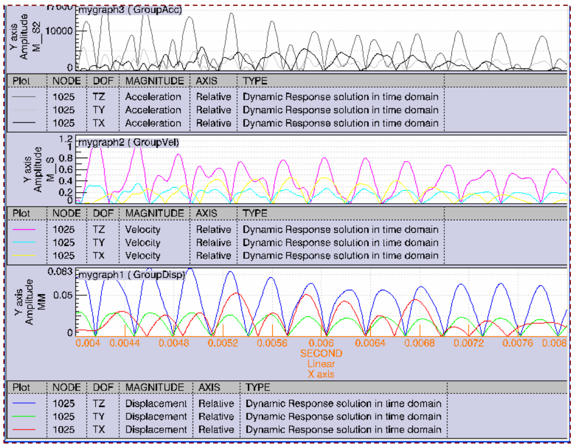
2 Graphs: Translations and Translational velocity

3 Graphs: Translations & Translational velocity & Acceleration

Acceleration

Velocity

Translation



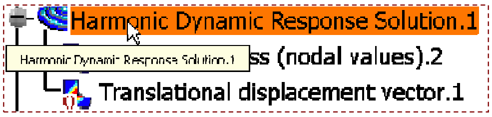
You can visualize the relative displacements, velocities or accelerations due to the restraint excitation.

Instructor Notes:

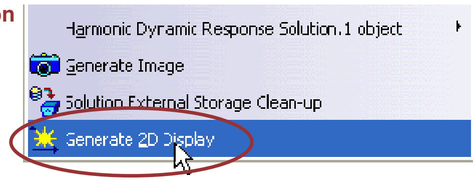
How to Generate a 2D Display (1/2)

You will see how to display 2D Results

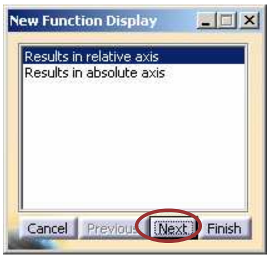
1 Right-click on



2 Click on

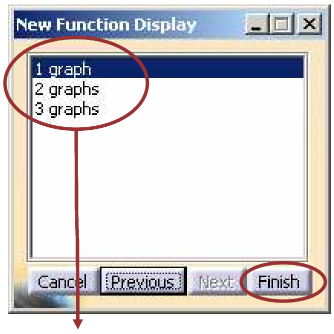


3 According to the dynamic case, you may have to choose between 2 types of axis



4 Click on next

5 Decide which output results you want to display



1 Graph: Translations

2 Graphs: Translations & translational velocity

3 Graphs: Translations & translational velocity & acceleration

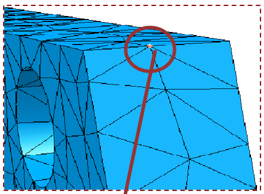
6 Click on Finish

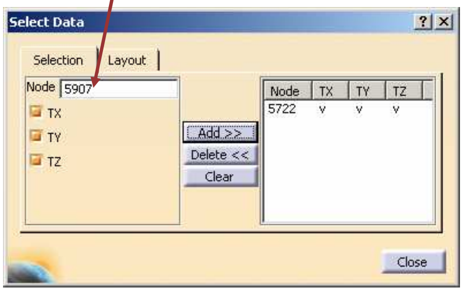
Instructor Notes:

How to Generate a 2D Display (2/2)

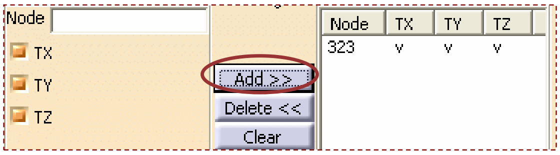
The "Select Data" dialog box appears. You have to specify a node .

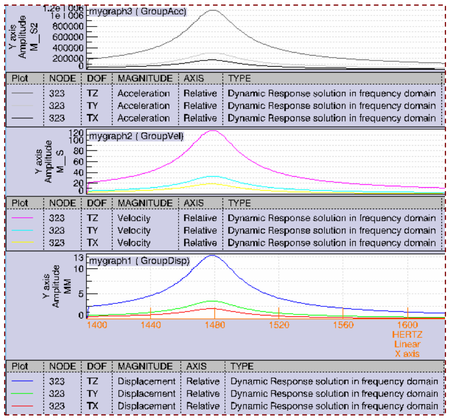
6 Select a node on your part/assembly, I.e 323





7 Click on add



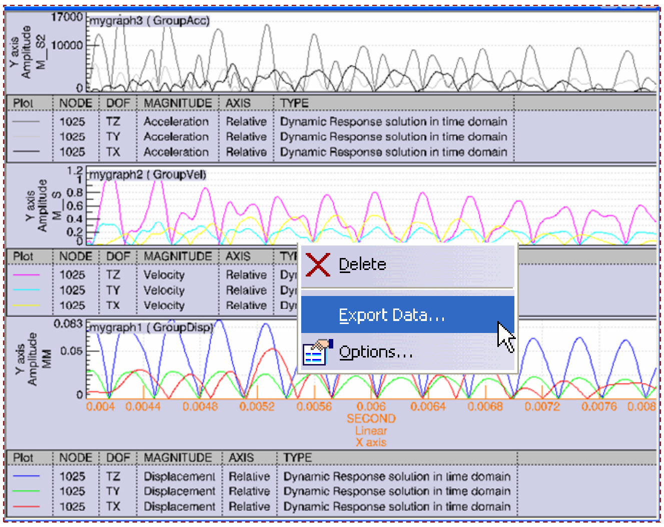


According to the case you have computed (harmonic or transient) you will get the amplitude versus Frequency or versus time. You can Double-click on the axis to customize.

Instructor Notes:

How to Export 2D Display Data

Once you have generated 2D results you can export curves Data



1. Right-click on a curve
2. Click on Export Data
3. Choose a format

Instructor Notes:

To Sum Up ...

In the lesson, you have seen two ways to generate images :

- Using the GPS tool
- Using the 2D Display GDY tool

Instructor Notes:

Master Exercise: Transient Dynamic Analysis

You will practice concepts learned throughout the course by building the master exercise and following the recommended process

- **Hood Analysis Presentation**
- **Transient Dynamic Analysis: Computing a Static Analysis on the Hood**
- **Transient Dynamic Analysis: Computing the Frequency Analysis on a Hood**
- **Transient Dynamic Analysis: Defining the Transient Dynamic Case on a Hood**

Instructor Notes:

Exercise

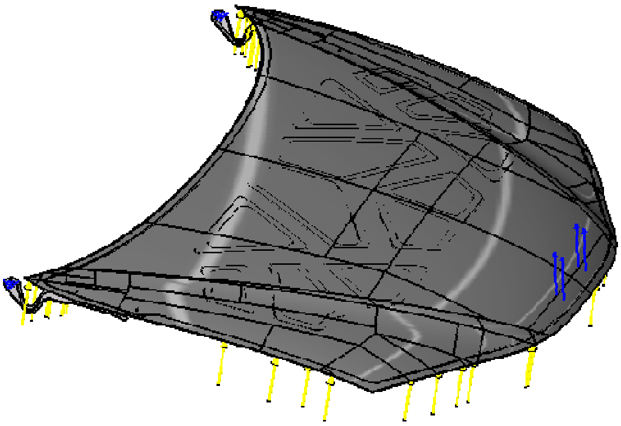
P2

Exercises marked with this callout will work in P2 configuration only

Transient Dynamic Analysis: Presentation

60 min

You will compute a Transient Dynamic Analysis to simulate a chock on a hood.
You will be able to visualize stress and displacements versus time.



Instructor Notes:

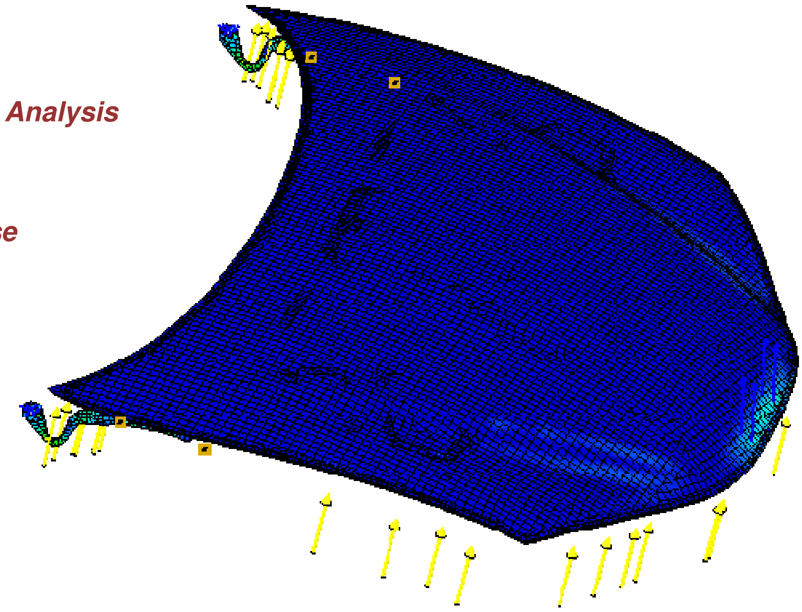
Transient Dynamic Analysis

Step 1 - Computing a Static Analysis on the hood

20 min.

In this step you will :

- Complete a Static Case Analysis
- Apply Loads
- Define Restraints
- Compute the Static Case



Instructor Notes:

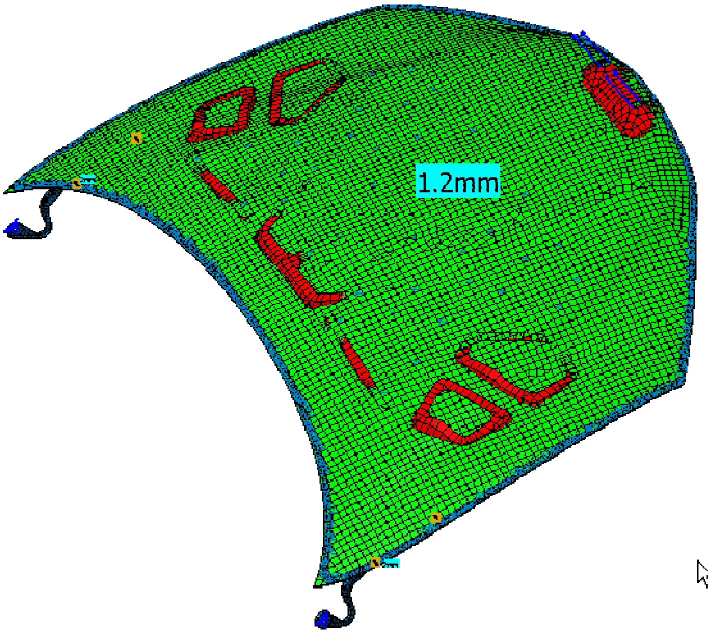
Transient Dynamic Analysis

Step 2 - Computing the Frequency Analysis on a hood

10 min.

In this step you will :

- *Define a Frequency Case*
- *Reuse the static restraints*



Instructor Notes:

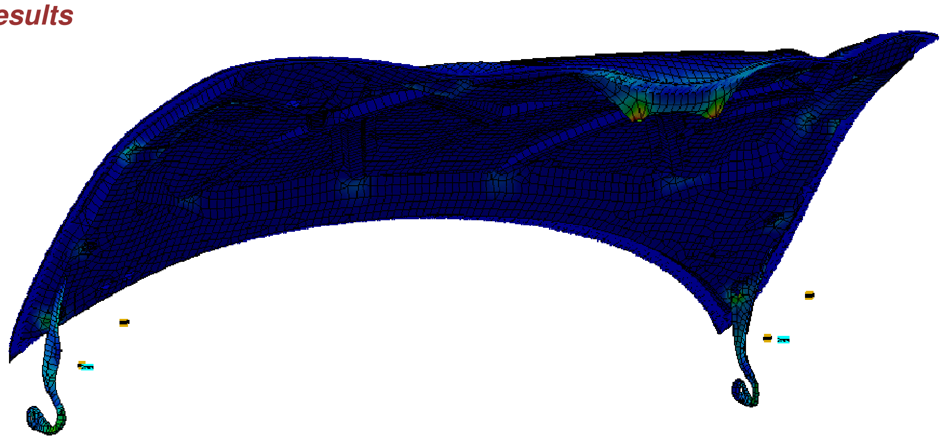
Transient Dynamic Analysis

Step 3 - Computing the Transient Dynamic Case

30 min.

In this step you will :

- *Define a Transient dynamic case using the static and frequency cases previously defined.*
- *Visualize the results*



Instructor Notes:

To Sum Up ...

In the course, you have seen how to define the different types of Dynamic Analysis :

- **Harmonic**
- **Transient**

And how to visualize the results.

Instructor Notes: