Abaqus/Dymola Co-Simulation: cantilever beam

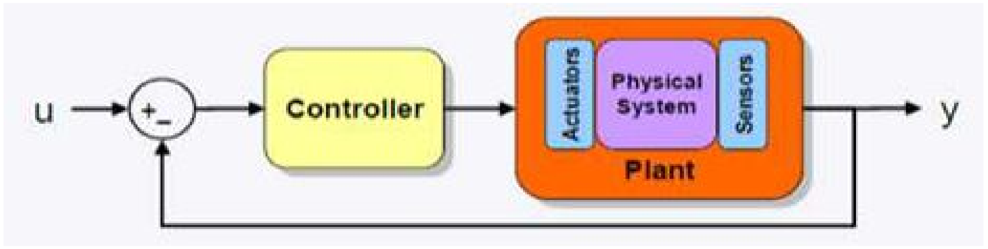
Author: Ahmed ELLEUCH
Angelo PALMIERI

Sources: Digital Product Simulation

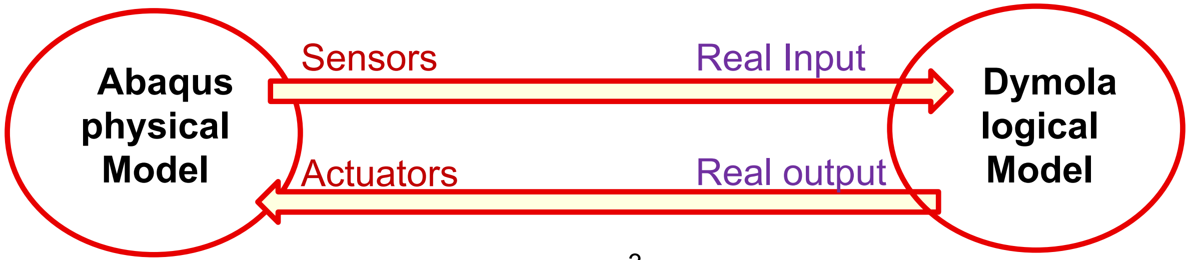
Co-Simulation

Introduction Co-Simulation

- The Abaqus co-simulation technique can be used to solve complex systems that include electronics such as control systems, electro-mechanics, hydraulics, and pneumatics by **coupling Abaqus with Dymola**, a general-purpose logical modeling software distributed by Dassault Systèmes.
- A **logical-physical** model looks as follows:



The communication between the two solvers is described schematically as shown below:



System requirements

- In this co-simulation we use this version:
  - Abaqus 6.14
  - Dymola 2015 (64-bit)

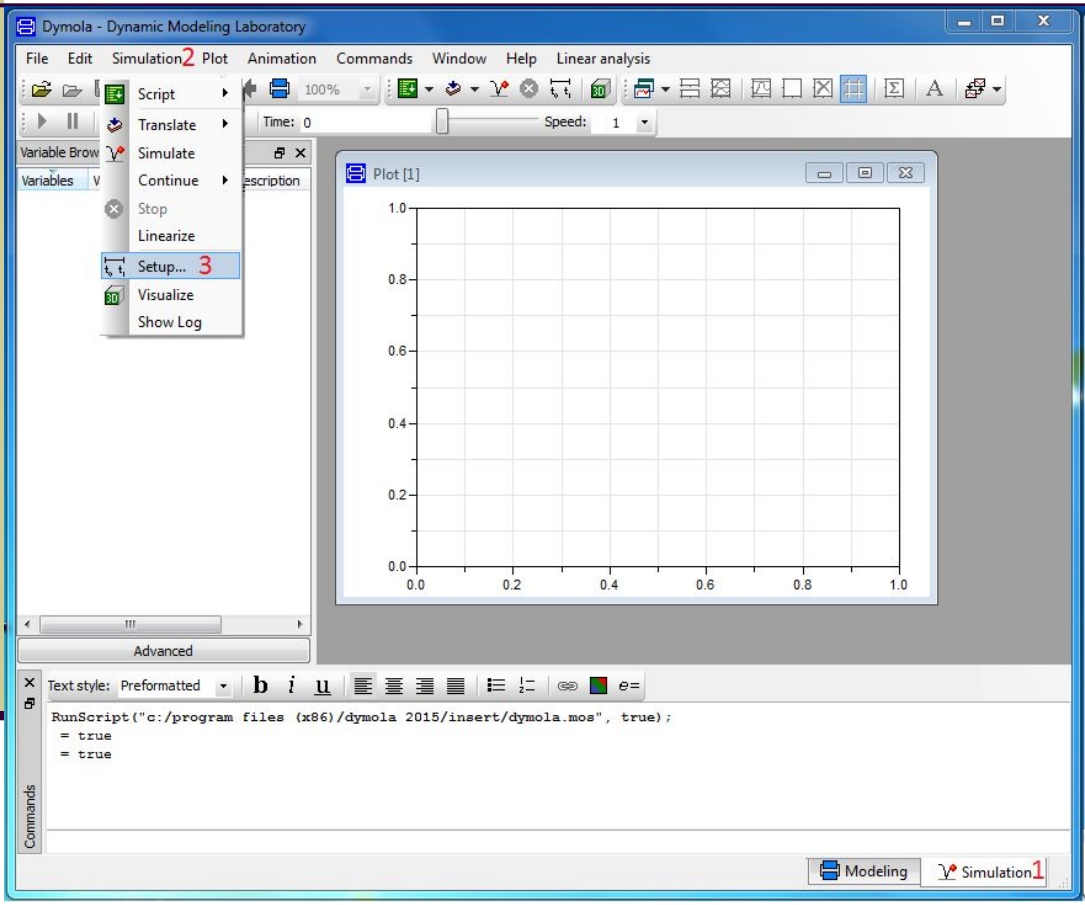
- The Dymola part of the co-simulation analysis must be run on a Windows 64-bit for Abaqus 6.14 platform.

- Verify the Dymola license:

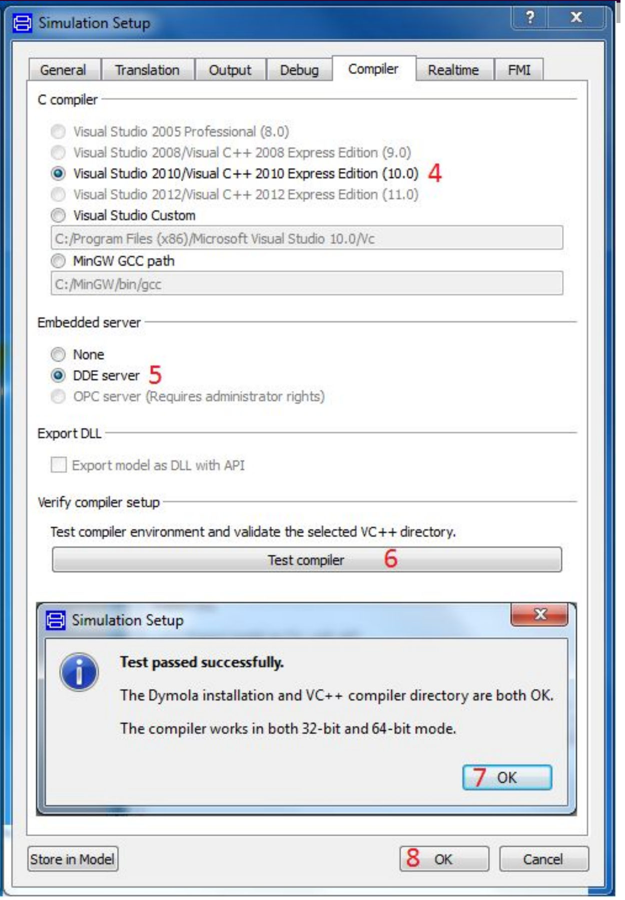
Launch Dymola, switch to the Simulation tab (bottom right of the GUI):

Go to **Simulation**>**Setup**>**Compiler**; the window on the right will pop up. Verify the compiler on your machine by clicking Verify. Make sure that you have the **Export DLL** option. If you do, check the button as shown later.

System requirements



```
RunScript("c:/program files (x86)/dymola 2015/insert/dymola.mos", true);
 = true
 = true
```



C compiler
- Visual Studio 2005 Professional (8.0)
- Visual Studio 2008/Visual C++ 2008 Express Edition (9.0)
- [✓ Visual Studio 2010/Visual C++ 2010 Express Edition (10.0)] 4
- Visual Studio 2012/Visual C++ 2012 Express Edition (11.0)
- Visual Studio Custom
  C:/Program Files (x86)/Microsoft Visual Studio 10.0/Vc
- MinGW GCC path
  C:/MinGW/bin/gcc

Embedded server
- None
- [✓ DDE server] 5
- OPC server (Requires administrator rights)

Export DLL
- Export model as DLL with API

Verify compiler setup

Test compiler environment and validate the selected VC++ directory.

Simulation Setup

Test passed successfully.

The Dymola installation and VC++ compiler directory are both OK.

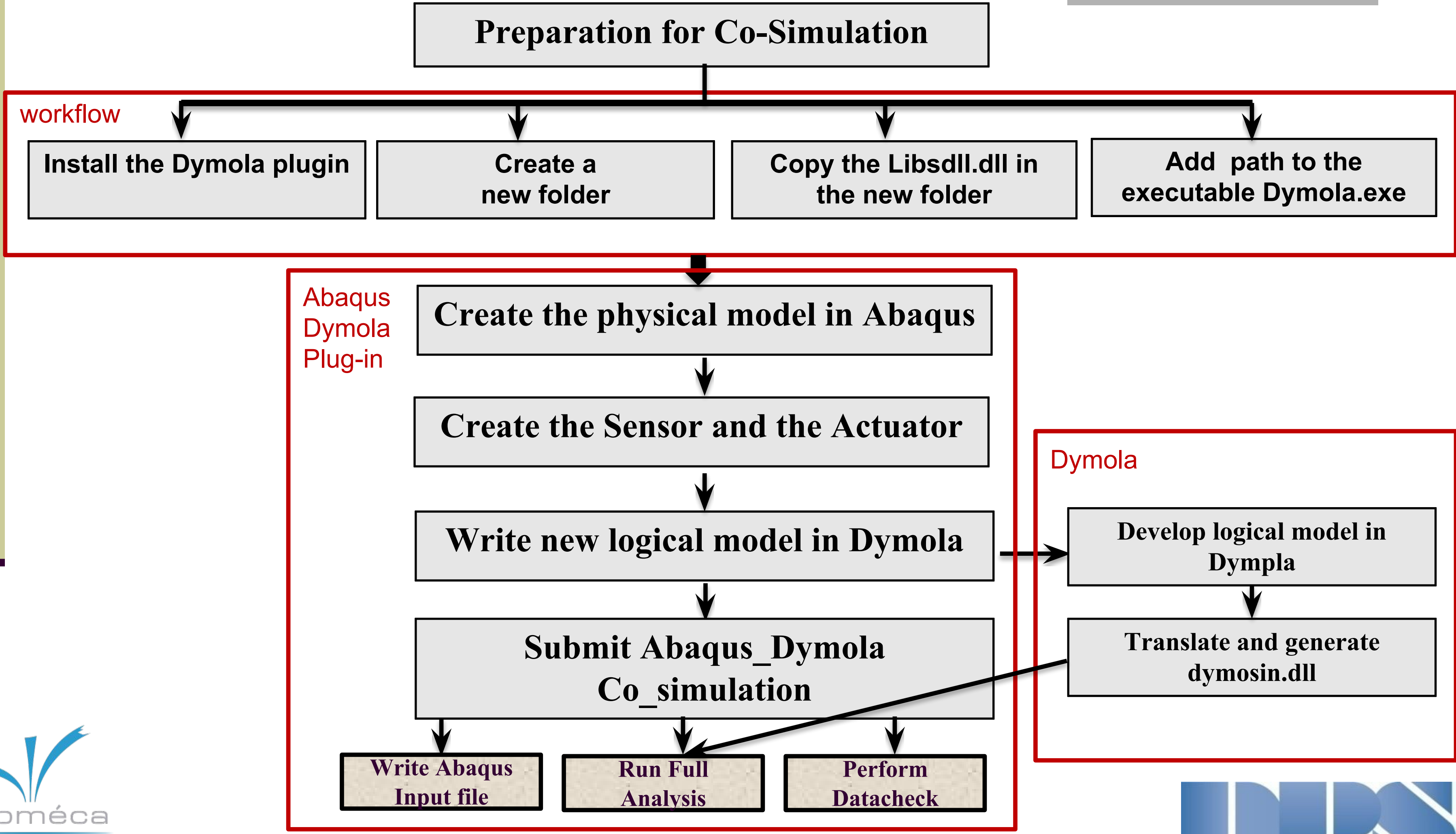
The compiler works in both 32-bit and 64-bit mode.

7 OK

Co-simulation objectives

- Couple Dymola 2015 and Abaqus 6.14.
- Develop a logical-physical modeling.
- Create a simple control system in Dymola for co-simulation.
- Run a co-simulation between Abaqus 6.14 and Dymola 2015.
- Review the co-simulation results.

Co-simulation



Preparation for Co-Simulation
workflow
Install the Dymola plugin
Create a new folder
Copy the Libsdll.dll in the new folder
Add path to the executable Dymola.exe
Abaqus Dymola Plug-in
Create the physical model in Abaqus
Create the Sensor and the Actuator
Write new logical model in Dymola
Submit Abaqus_Dymola Co_simulation
Write Abaqus Input file
Run Full Analysis
Perform Datacheck
Dymola
Develop logical model in Dympla
Translate and generate dymosin.dll

Co-simulation

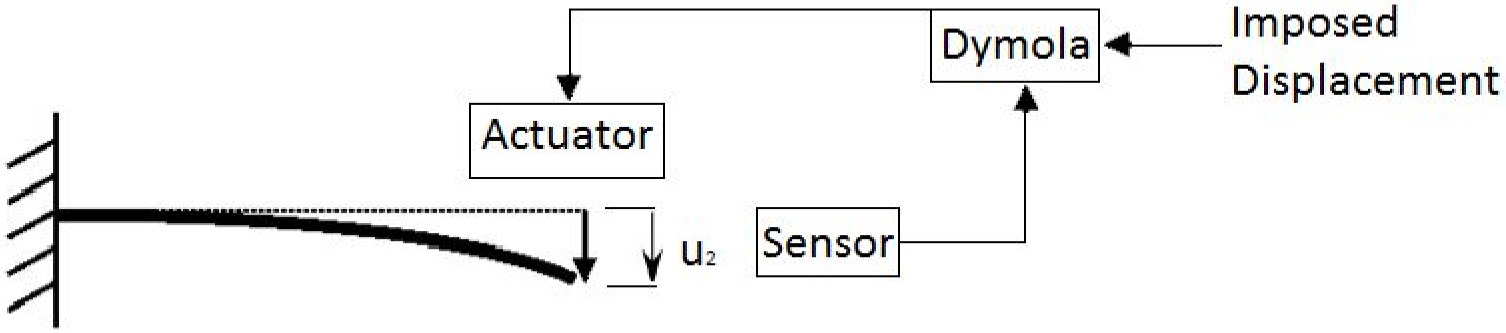
- **Plug-in:** The plug-in provides the interface to create or identify sensors or actuators in Abaqus/CAE that can then be written to a new Dymola logical (*.mo) file or can be matched with an existing logical file.
- **Dymosim.dll:** This file will be create when we simulate a model in Dymola,

If you work with an **educative license the file ***dymosim.dll*** generated by the translation process can't be compatible with the Co-simulation process .The correct ***dymosim.dll*** *file could be generated only from an* ***industrial license*** *of Dymola. Use so the file "dymosim.dll " from the industrial license and replace it.*

If you work with* ***industrial license *so this file will be created automatically.*

Co-simulation: examples

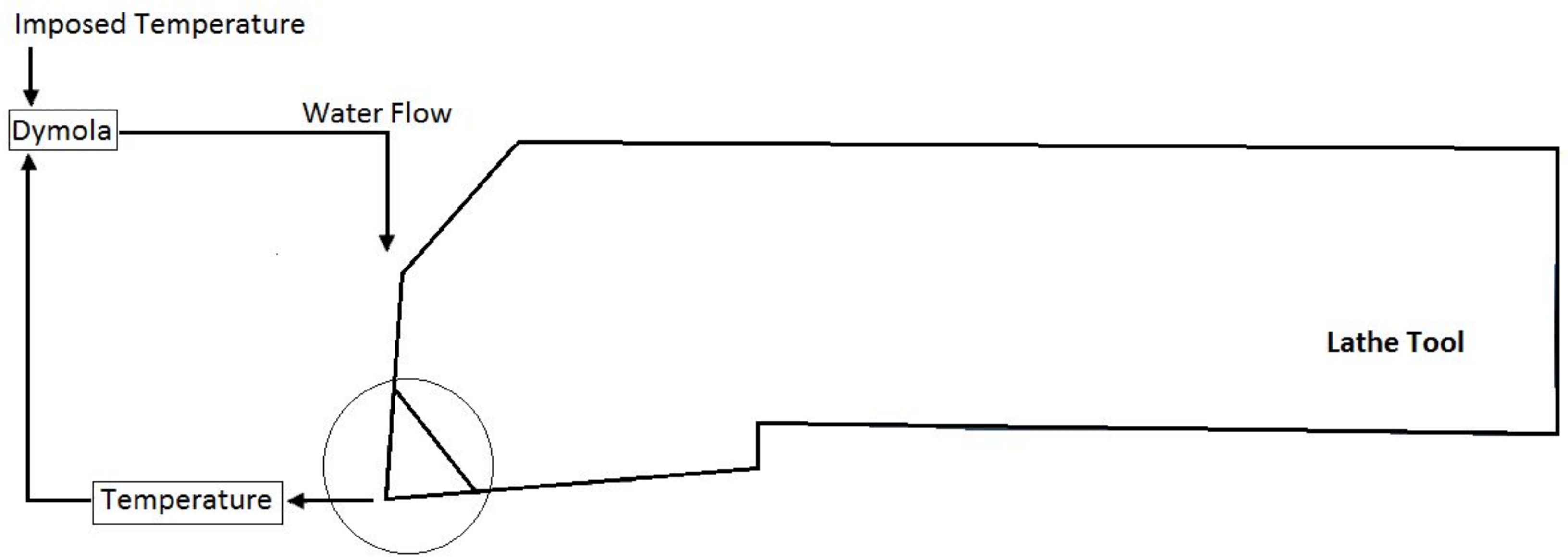
Scheme of Co-simulation for a Beam:



In this Scheme we will make a sensor on the beam that sends the position to Dymola and receive an amplitude pressure (Actuator) to reach some points in function of the time.

Co-simulation: examples

Scheme of Co-simulation for a Lathe Tool:

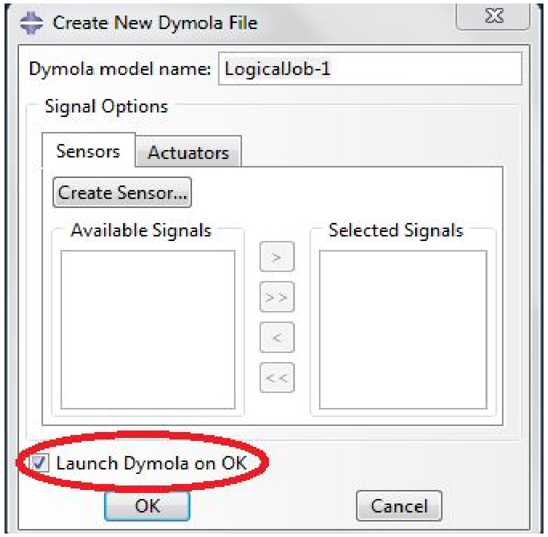


In this example we will create a three dimensional **lathe tool** and we simulate the heat produced by the friction, control the flux of water cooling down the tool in order to maintain a target temperature at the tip.

Preparation for Co-Simulation

Preparation for the co-simulation:

- **Objectif:** Add the Path of Dymola:
- The complete path to the executable that launches Dymola, Dymola.exe, should be added to the environment variable PATH. On Windows machines, a permanent setting can be employed by modifying the system or user environment variable.(To launch Dymola from Abaqus when you whrite a new Dymola file).


Create New Dymola File

Dymola model name: LogicalJob-1

Signal Options

Sensors | Actuators

Create Sensor...

Available Signals

Selected Signals

[✓] Launch Dymola on OK

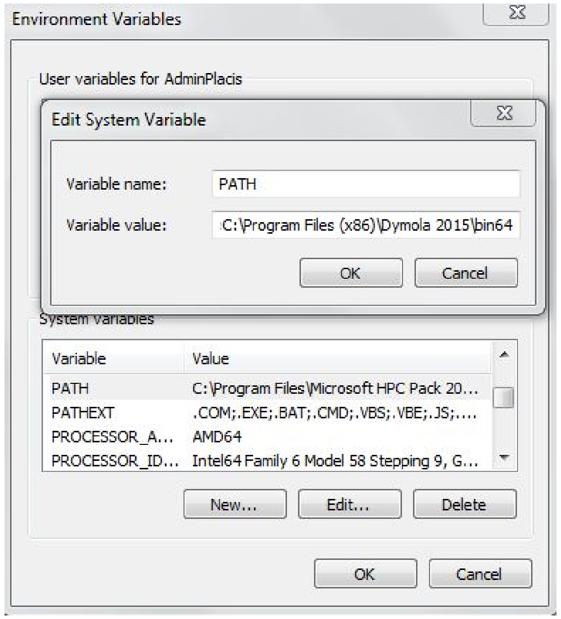
OK   Cancel

Preparation for the co-simulation:

1- From the **Control Panel\System and Security\System**, click on the **Advanced** tab and then click **Environment Variables**.

2- Click New and enter PATH for the variable name and specify the path to the Dymola executable (“C:\Program Files (x86)\Dymola 2015\bin64”) for the variable value. If there is an existing PATH variable, edit the variable and add the path to the executable to the variable value.

3- Click OK and OK.



Variable name: PATH

Variable value: C:\Program Files (x86)\Dymola 2015\bin64

Variable	Value
PATH	C:\Program Files\Microsoft HPC Pack 20...
PATHEXT	.COM;.EXE;.BAT;.CMD;.VBS;.VBE;.JS;....
PROCESSOR_A...	AMD64
PROCESSOR_ID...	Intel64 Family 6 Model 58 Stepping 9, G...

supméca

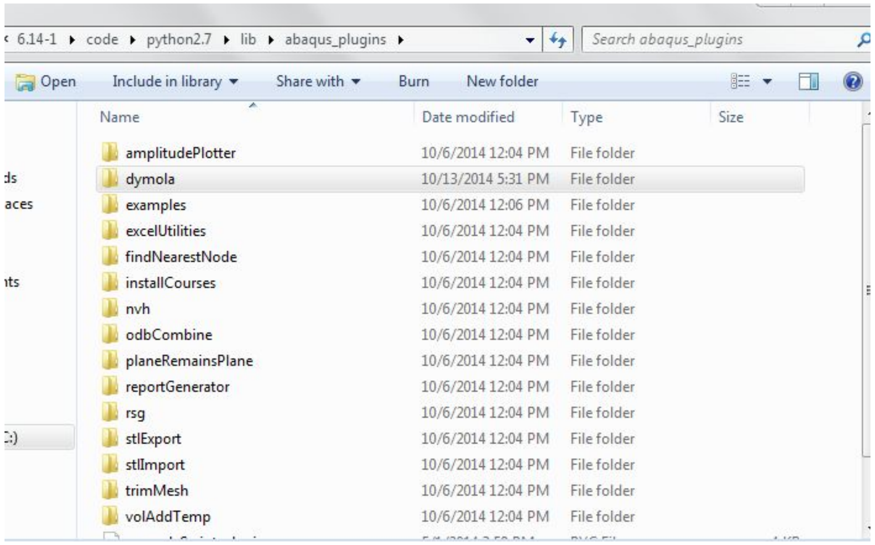
Digital Product Simulation

Preparation for the co-simulation:

- To install the plug-in, you need to do the following:

1- Use the plugin. It’s named «Dymola_plugin»

2- You have to copy the content of this plugin file on the abaqus 6.14 directory : C:\SIMULIA\Abaqus\6.14-1\code\python2.7\lib\abaqus_plugins



6.14-1 ▸ code ▸ python2.7 ▸ lib ▸ abaqus_plugins ▸

Search abaqus_plugins

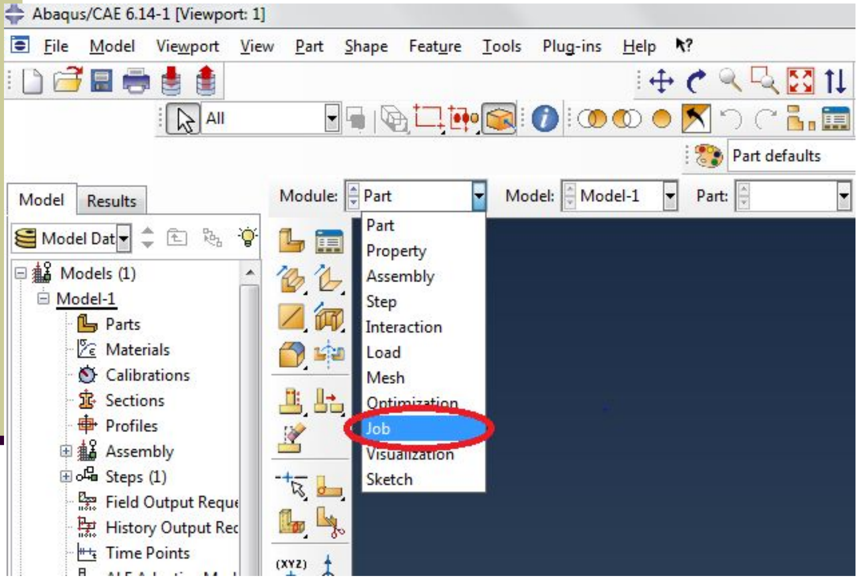
Name	Date modified	Type	Size
amplitudePlotter	10/6/2014 12:04 PM	File folder	
dymola	10/13/2014 5:31 PM	File folder	
examples	10/6/2014 12:06 PM	File folder	
excelUtilities	10/6/2014 12:04 PM	File folder	
findNearestNode	10/6/2014 12:04 PM	File folder	
installCourses	10/6/2014 12:04 PM	File folder	
nvh	10/6/2014 12:04 PM	File folder	
odbCombine	10/6/2014 12:04 PM	File folder	
planeRemainsPlane	10/6/2014 12:04 PM	File folder	
reportGenerator	10/6/2014 12:04 PM	File folder	
rsg	10/6/2014 12:04 PM	File folder	
stlExport	10/6/2014 12:04 PM	File folder	
stlImport	10/6/2014 12:04 PM	File folder	
trimMesh	10/6/2014 12:04 PM	File folder	
volAddTemp	10/6/2014 12:04 PM	File folder	

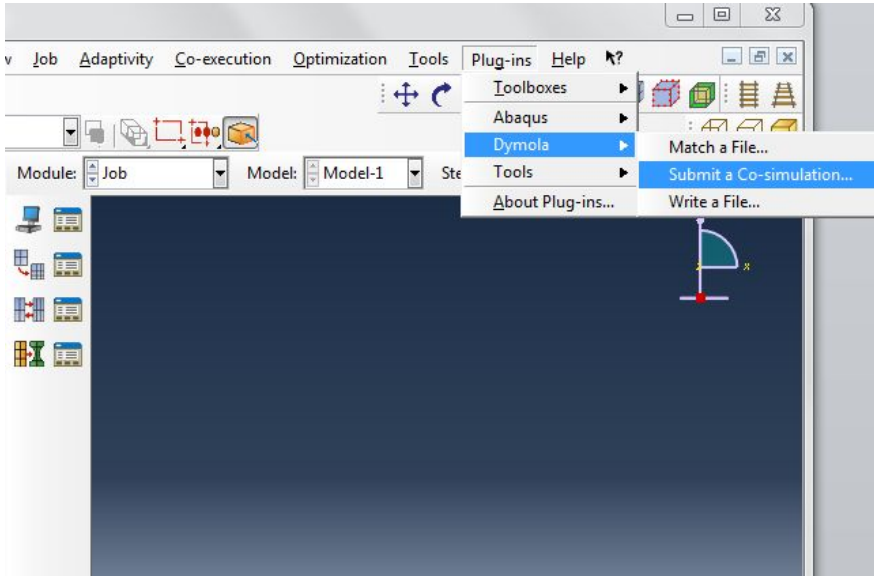
supméca

Digital Product Simulation

Preparation for the co-simulation:

2- Then you can open the Abaqus 6.14 and verify if the plugin is installed by going to **JOB** section and see under the "**plugins**" toolbox if there is the Dymola plugin.



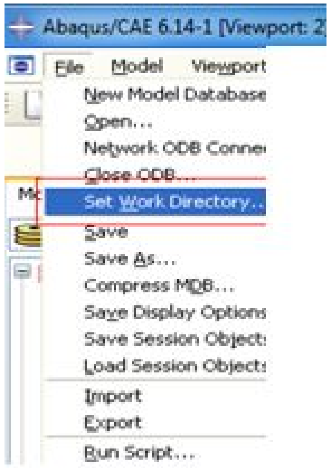


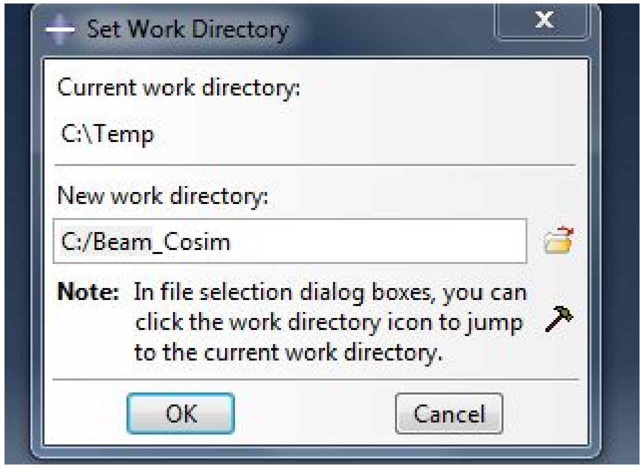
Digital Product Simulation

Preparation for the co-simulation:

3- Once you have made this verification create a new file on the C:\ and name it "Beam Co-simulation" .

4-Set this file as the work directory of Abaqus .





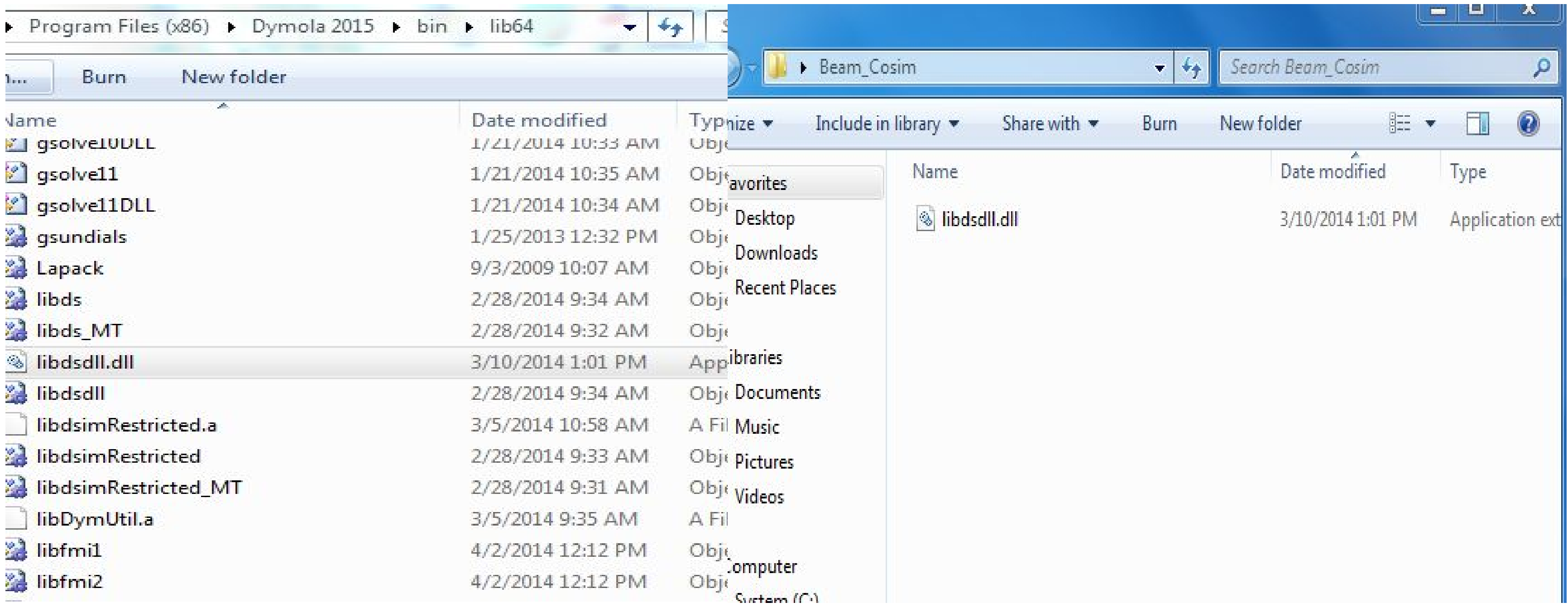
Current work directory:
C:\Temp

New work directory:
C:/Beam_Cosim

Note: In file selection dialog boxes, you can click the work directory icon to jump to the current work directory.

Preparation for the co-simulation:

5-Copy the files ”libsdll.dll” from “C:\Program Files (x86)\Dymola 2015\bin\lib64” to the “Beam co-simulation” file.



Name	Date modified	Typ
gsolve10DLL	1/21/2014 10:33 AM	Obj
gsolve11	1/21/2014 10:35 AM	Obj
gsolve11DLL	1/21/2014 10:34 AM	Obj
gsundials	1/25/2013 12:32 PM	Obj
Lapack	9/3/2009 10:07 AM	Obj
libds	2/28/2014 9:34 AM	Obj
libds_MT	2/28/2014 9:32 AM	Obj
libdsdll.dll	3/10/2014 1:01 PM	App
libdsdll	2/28/2014 9:34 AM	Obj
libdsimRestricted.a	3/5/2014 10:58 AM	A Fil
libdsimRestricted	2/28/2014 9:33 AM	Obj
libdsimRestricted_MT	2/28/2014 9:31 AM	Obj
libDymUtil.a	3/5/2014 9:35 AM	A Fil
libfmi1	4/2/2014 12:12 PM	Obj
libfmi2	4/2/2014 12:12 PM	Obj

Name	Date modified	Type
libdsdll.dll	3/10/2014 1:01 PM	Application ext

Digital Product Simulation

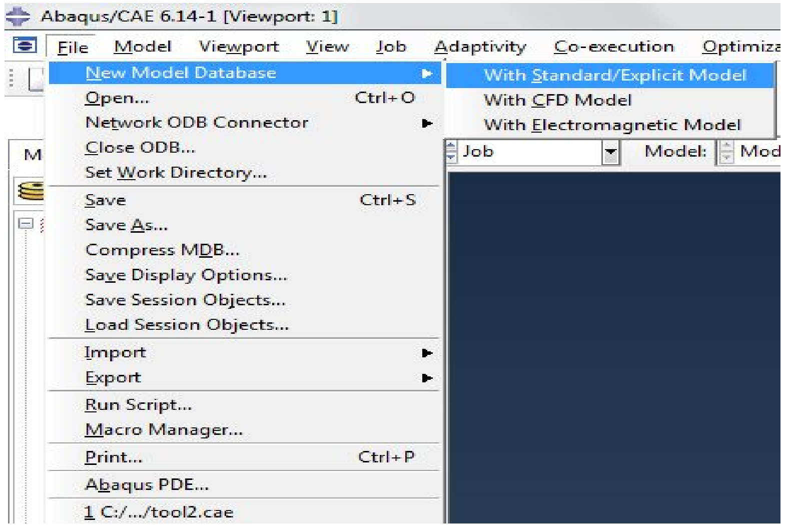
Creation of the Abaqus model

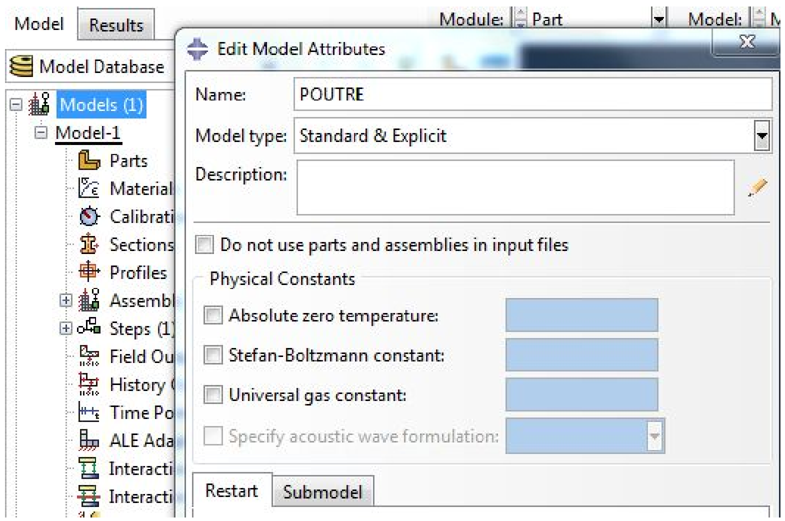
Creation of the Abaqus model

1- Click **file->new model database->with standard explicit model**.

2- In the construction tree, double-click **Models (1)** to create a new model in the database. The editing model attributes window appears.

3- In the Edit Model Attributes window, enter name as **POUTRE** and click OK.





Edit Model Attributes

Name: POUTRE

Model type: Standard & Explicit

Description:

☐ Do not use parts and assemblies in input files

Physical Constants

☐ Absolute zero temperature:

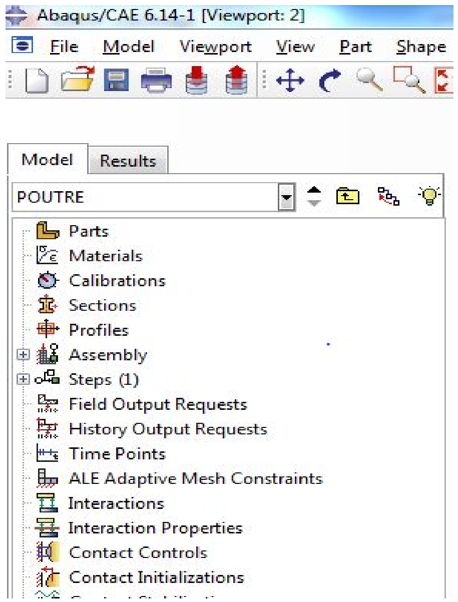
☐ Stefan-Boltzmann constant:

☐ Universal gas constant:

☐ Specify acoustic wave formulation:

Creation of the Abaqus model

4- Select poutre as "root" (**right click at POUTRE and select Set as Root menu that appears**). The tree is then built as shown in Figure.



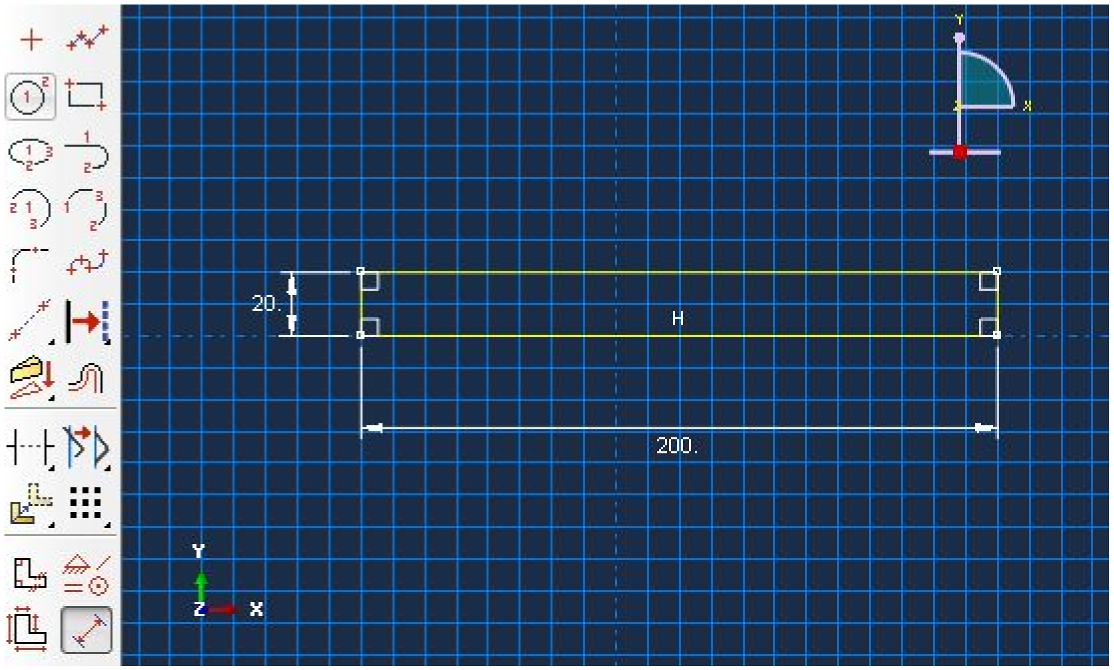
5- Save the database, select **File** → **Save** at the main menu and select "**Beam co-simulation**" as the database.
Click OK. The .cae is added automatically in the **Beam Co-simulation file.**

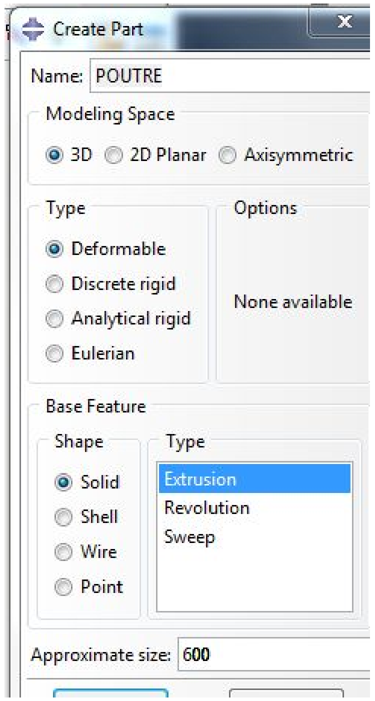
**In this section, you will create a deformable solid 3D drawing of the beam profile in 2D (a rectangle) and then extruding.

1- In the construction tree, double-click **Parts** to create a new share in the beam model. Create the Share window appears.

Creation of the Abaqus model

2- In this window, enter the name as **POUTRE** and specify an **approximate size of 600** Accept the default settings. **3D Deformable Solid, Extrude**. Click **Continue**.





Name: POUTRE

Modeling Space: [✓ 3D] 2D Planar Axisymmetric

Type: [✓ Deformable] Discrete rigid Analytical rigid Eulerian

Options: None available

Base Feature — Shape: [✓ Solid] Shell Wire Point

Type: [✓ Extrusion] Revolution Sweep

Approximate size: 600

Digital Product Simulation

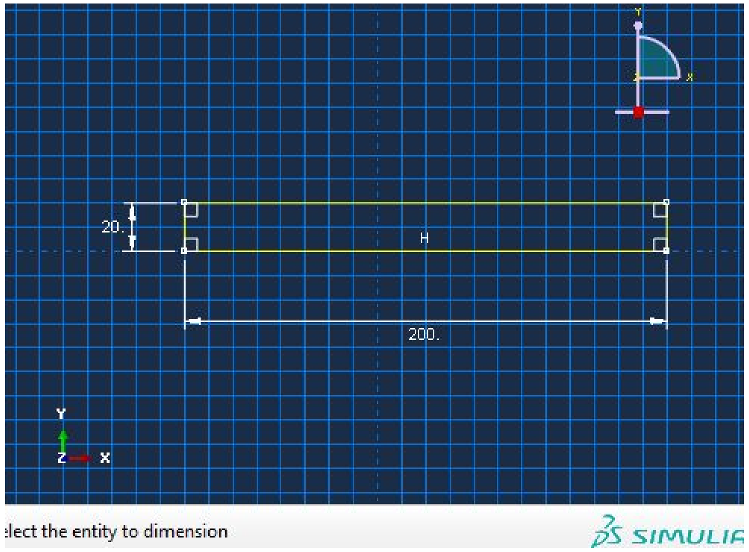
Creation of the Abaqus model

Draw the rectangular profile:

1- Click the Create tool Lines: Rectangle appears in the upper right of the toolbar.

2- Use the **Add Dimension** to define the dimensions of the top and left sides of the rectangle.

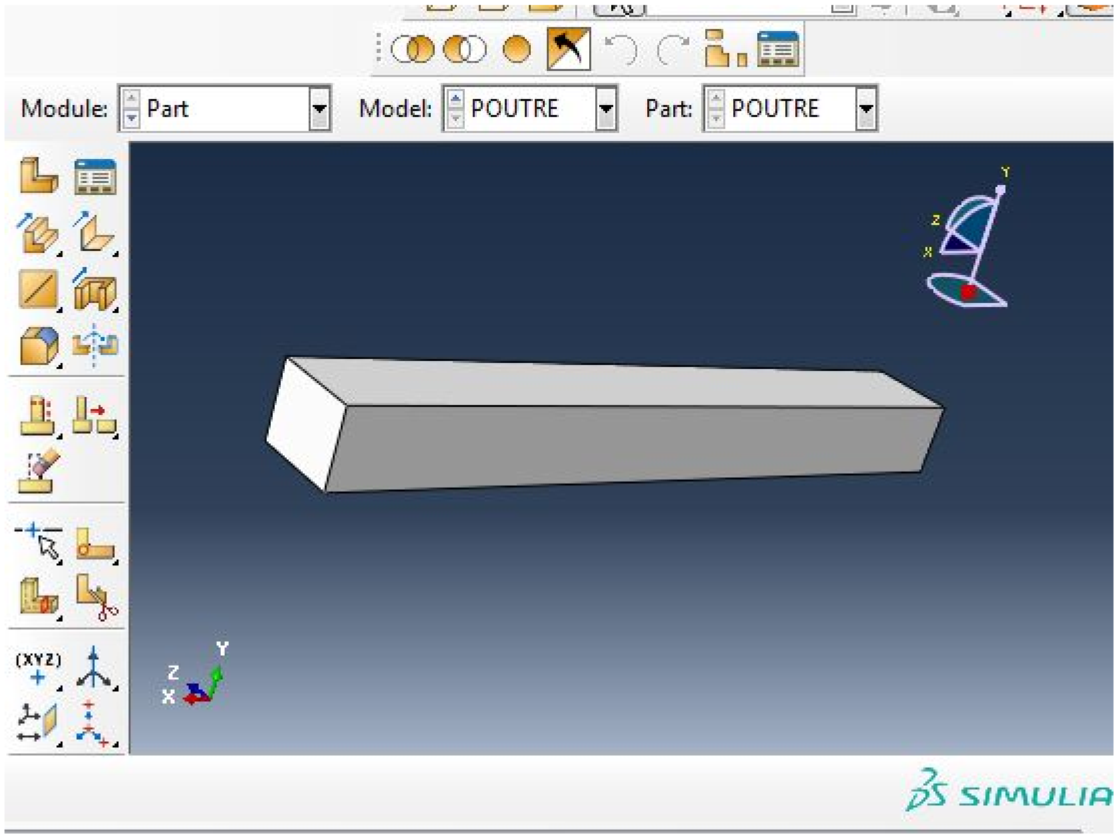
The upper side must have a horizontal dimension of **200 mm** and the left side must have a vertical dimension of **20 mm**.



elect the entity to dimension

Creation of the Abaqus model

3- Click **Done** or the middle mouse button. Depth of the **extrusion 25** in the Edit Base Extrusion window, then click OK.



Creation of the Abaqus model

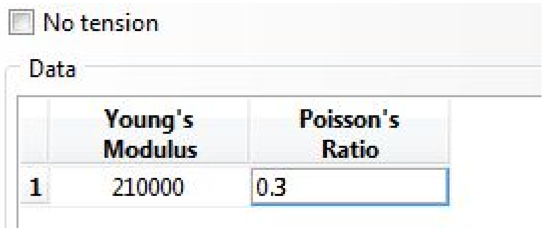
**Create a linear elastic material with a Young's modulus of 210 000 MPa and Poisson's ratio of 0.3

1- In the construction tree, double-click **Materials** to create a new material.

2- Edit Material window will open, name the material: **Steel**.

3- Starting material menu, select **Mechanical** → **Elasticity** → **Elastic**.

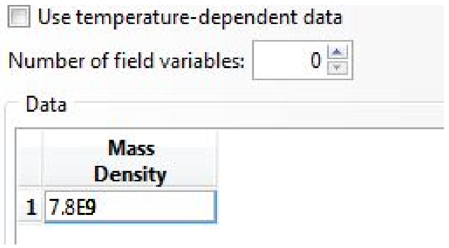
4- Select **General** → **Density** and enter a density **of 7.8 E9 tonnes/mm3**. Click OK



☐ No tension

Data

	Young's Modulus	Poisson's Ratio
1	210000	0.3



☐ Use temperature-dependent data

Number of field variables: 0

Data

	Mass Density
1	7.8E9

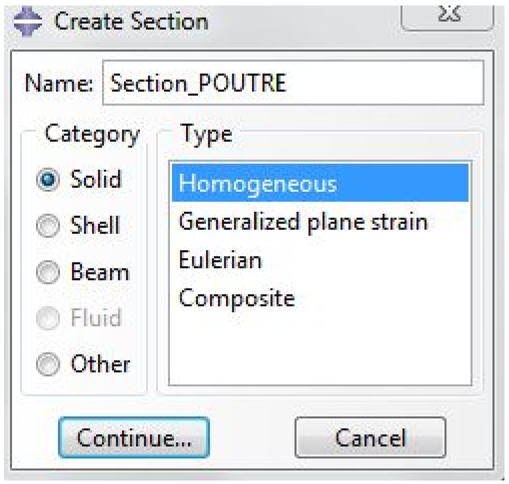
Creation of the Abaqus model

**The next step is the creation of a solid section property and its allocation to the POUTRE. Section will refer to the steel material that you just created.

1- In the construction tree, double-click **Sections** to create a new section.

2- In this window: - Name the section **Section_POUTRE**

- Accept the category and type default: Solid and Homogenous.
- Click **Continue**


Create Section

Name: Section_POUTRE

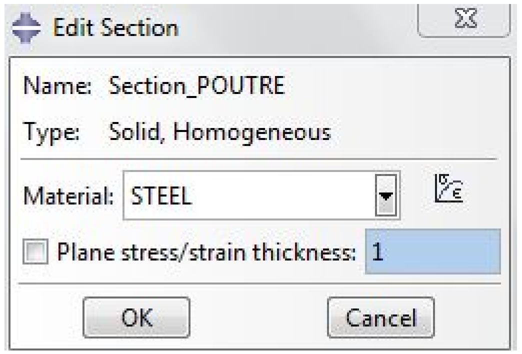
Category: [✓ Solid] Shell Beam Fluid Other

Type: [✓ Homogeneous] Generalized plane strain Eulerian Composite

Creation of the Abaqus model

3- New window appears (**Edit Section**):

- Choose as Material **Steel**
- Click OK



Name: Section_POUTRE

Type: Solid, Homogeneous

Material: STEEL

☐ Plane stress/strain thickness: 1

**The next step is the ceation of section assignement:

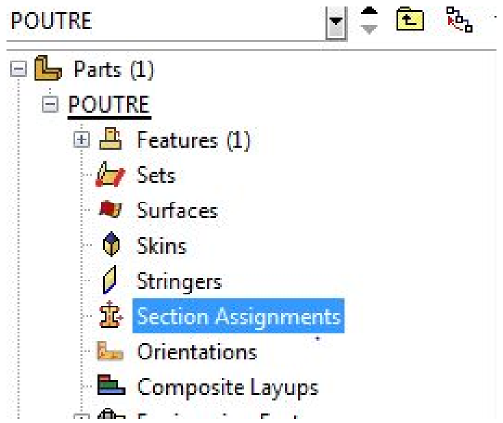
1- In the tree construction, double-click **Section Assignments** to assign a section to the beam part.

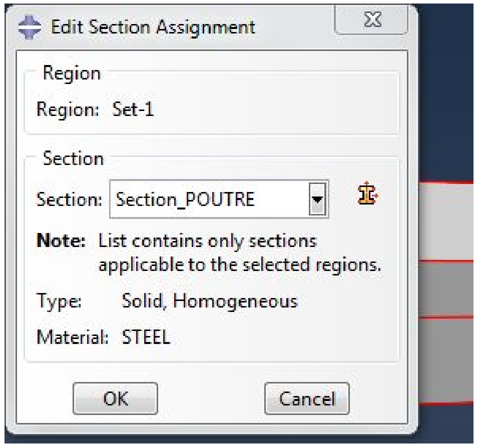
2- Click anywhere on the beam to select the part.

Creation of the Abaqus model

3- Click **Done** in the prompt area. The Edit Section Assignment window appears.

Then click Ok.




Edit Section Assignment

Region

Region: Set-1

Section

Section: Section_POUTRE

Note: List contains only sections applicable to the selected regions.

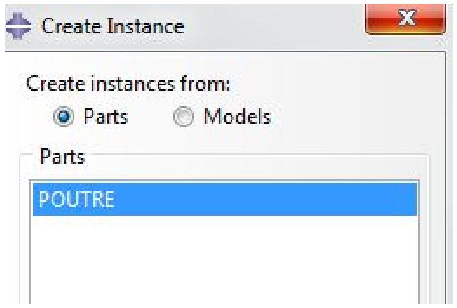
Type: Solid, Homogeneous

Material: STEEL

Now create the **Assembly Model**:

1- In the construction tree, double-click **Instances**

2- Click Ok


Create Instance

Create instances from: [✓ Parts] Models

Parts

POUTRE

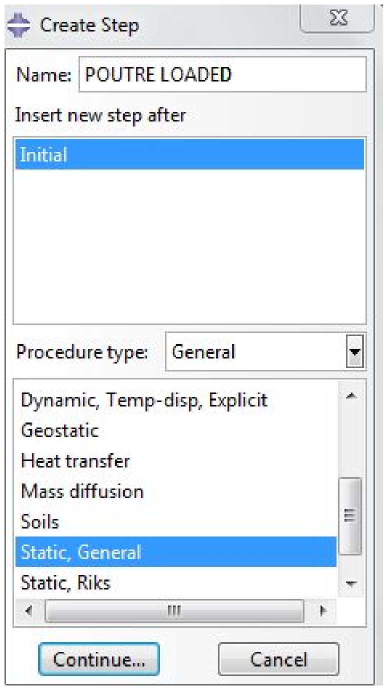
Creation of the Abaqus model

After the creation of the Assembly Model, we create **step

1- Double-click **Steps,** new window appear

2- In this window:

- Name the step **POUTRE LOADED**.
- From the list of available procedures in the Create Step window, select **Static, General**.
- Click Continue.



Create Step

Name: POUTRE LOADED

Insert new step after

Initial

Procedure type: General

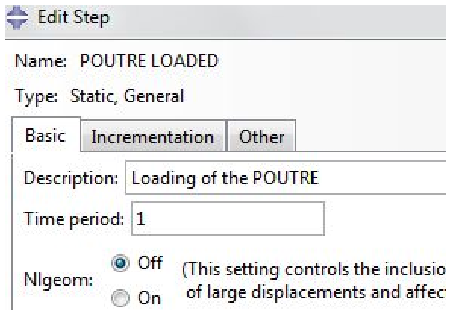
Dynamic, Temp-disp, Explicit
Geostatic
Heat transfer
Mass diffusion
Soils
Static, General
Static, Riks

Creation of the Abaqus model

**The editing step window appears:

1- At the Description field on the Basic tab, enter Charge poutre.

2- Set a value of **0.1** as an initial increment size.



Edit Step

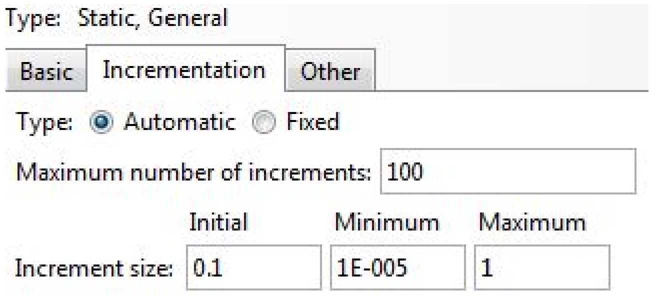
Name: POUTRE LOADED

Type: Static, General

Description: Loading of the POUTRE

Time period: 1

Nlgeom: [✓ Off] On (This setting controls the inclusio of large displacements and affec



Type: Static, General

Type: [✓ Automatic] Fixed

Maximum number of increments: 100

	Initial	Minimum	Maximum
Increment size:	0.1	1E-005	1

Creation of the Abaqus model

**CREATE THE APPLICATION OF BOUNDARY CONDITIONS TO AN END OF THE BEAM:

1- In the construction tree, double-click **BCs** to create a new boundary condition.

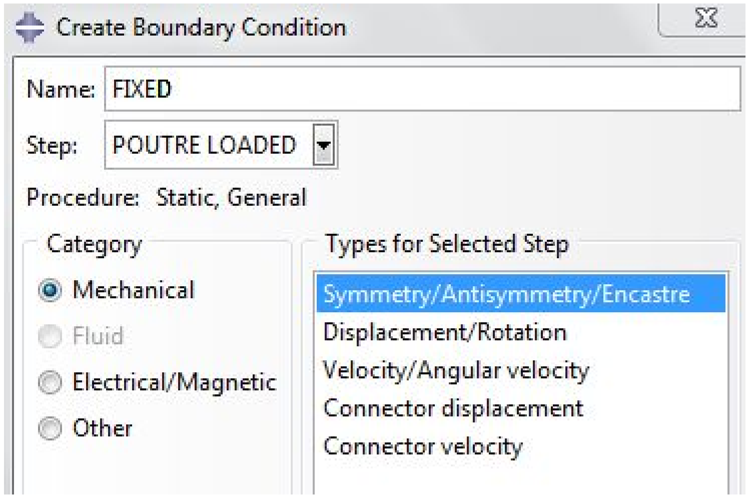
2- In this window:

- Name the boundary condition **Fixed**.
- Choose the step **POUTRE LOADED** in which the boundary condition will be activated.
- At the Category list, accept the default choice: Mechanical.
- Click Continue.

Creation of the Abaqus model

3- Select the face you want and click OK.

4- The Edit Boundary Condition window appears. Choose **ENCASTRED** to block all degrees of freedom and then click OK.



Create Boundary Condition

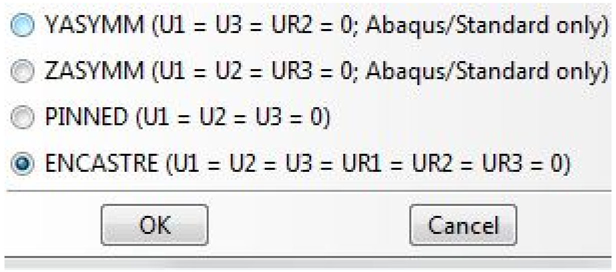
Name: FIXED

Step: POUTRE LOADED

Procedure: Static, General

Category: [✓ Mechanical] Fluid Electrical/Magnetic Other

Types for Selected Step: [✓ Symmetry/Antisymmetry/Encastre] Displacement/Rotation Velocity/Angular velocity Connector displacement Connector velocity

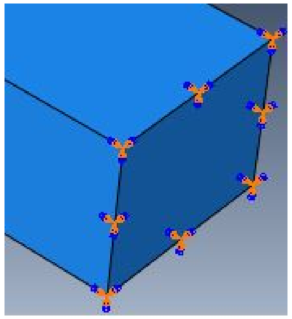


YASYMM (U1 = U3 = UR2 = 0; Abaqus/Standard only)

ZASYMM (U1 = U2 = UR3 = 0; Abaqus/Standard only)

PINNED (U1 = U2 = U3 = 0)

[✓ ENCASTRE (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0)]



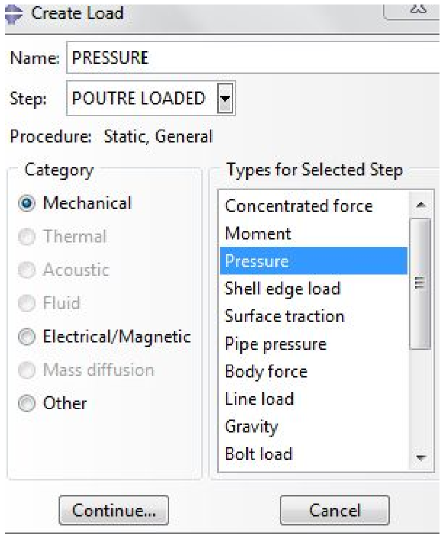
Creation of the Abaqus model

**The next step is: *APPLYING PRESSURE ON THE UPPER SURFACE OF THE BEAM*

1- Double-click the **Loads** container to create a new load

2- In this window:

- Name the load **PRESSURE**.
- Select Beam loaded as the load step at which will be applied.
- For the Category list, accept the default choice: **Mechanical**.
- At the Types for Selected Step list, select **Pressure**.
- Click Continue.



Name: PRESSURE
Step: POUTRE LOADED
Procedure: Static, General

Category: [✓ Mechanical] Thermal, Acoustic, Fluid, Electrical/Magnetic, Mass diffusion, Other

Types for Selected Step: Concentrated force, Moment, [✓ Pressure], Shell edge load, Surface traction, Pipe pressure, Body force, Line load, Gravity, Bolt load

supméca

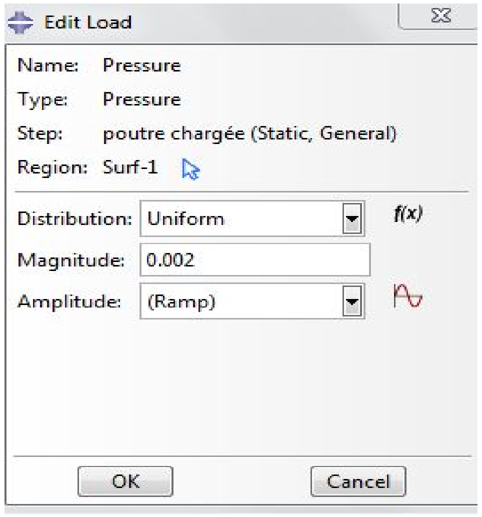
Digital Product Simulation

Creation of the Abaqus model

3- In the graphics window, select the top face as the face which the load is Applied.

4- In the Edit Load window:

- Enter a magnitude of **0.002** for loading
- Accept the default selections for options and Amplitude Distribution.
- Click OK



Name: Pressure
Type: Pressure
Step: poutre chargée (Static, General)
Region: Surf-1

Distribution: Uniform
Magnitude: 0.002
Amplitude: (Ramp)

supméca

Digital Product Simulation

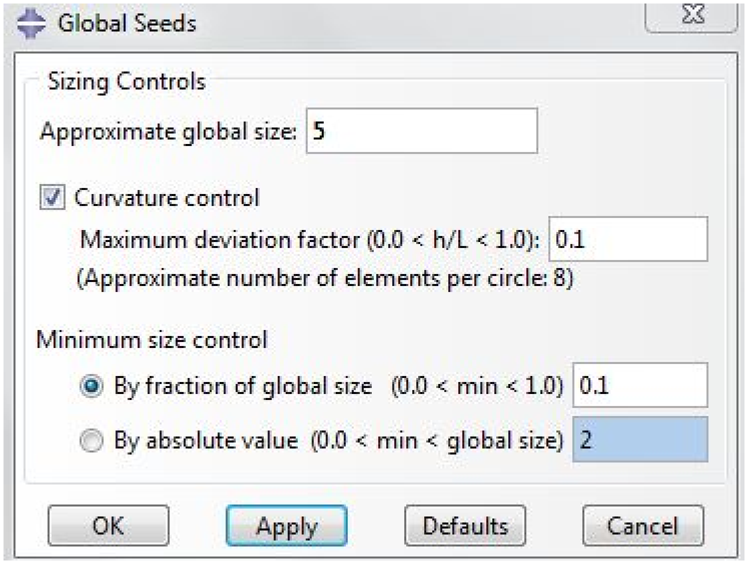
Creation of the Abaqus model

**After that we create the mesh:

1- Double-click the **Mesh** container in the tree under the POUTRE part.

2- At the main menu bar, select **Seed** → **Part** to set the overall density of the mesh.

3- In this dialog, enter an approximate overall size of **5** and click OK.


Global Seeds

Sizing Controls

Approximate global size: 5

[✓] Curvature control

Maximum deviation factor (0.0 < h/L < 1.0): 0.1

(Approximate number of elements per circle: 8)

Minimum size control

● By fraction of global size (0.0 < min < 1.0) 0.1

○ By absolute value (0.0 < min < global size) 2

Creation of the Abaqus model

Now we allocate **the control Mesh:

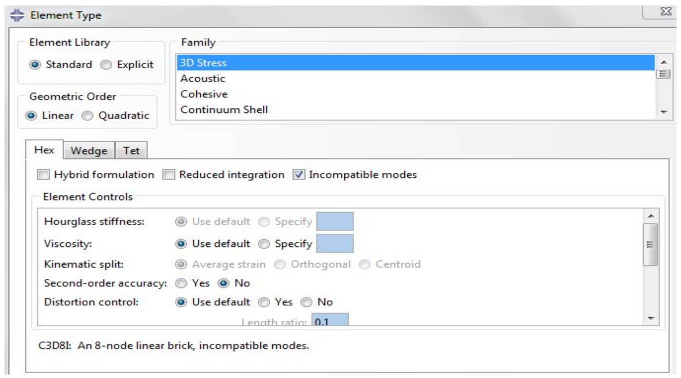
1- At the main menu, choose **Mesh** → **Controls**.

2- In the window Mesh Controls dialog box, accept **Hex**.

3- Click OK.

Next step: ASSIGNMENT OF A **TYPE OF ELEMENT**

1- At the main menu, select **Mesh** → **Element Type**.
2- In the Element Type window, accept the following default selections.



Element Library: [✓ Standard] Explicit

Geometric Order: [✓ Linear] Quadratic

Family: [✓ 3D Stress], Acoustic, Cohesive, Continuum Shell

Hex: Hybrid formulation, Reduced integration, [✓ Incompatible modes]

Element Controls

Hourglass stiffness: [✓ Use default] Specify

Viscosity: [✓ Use default] Specify

Kinematic split: [✓ Average strain] Orthogonal Centroid

Second-order accuracy: Yes [✓ No]

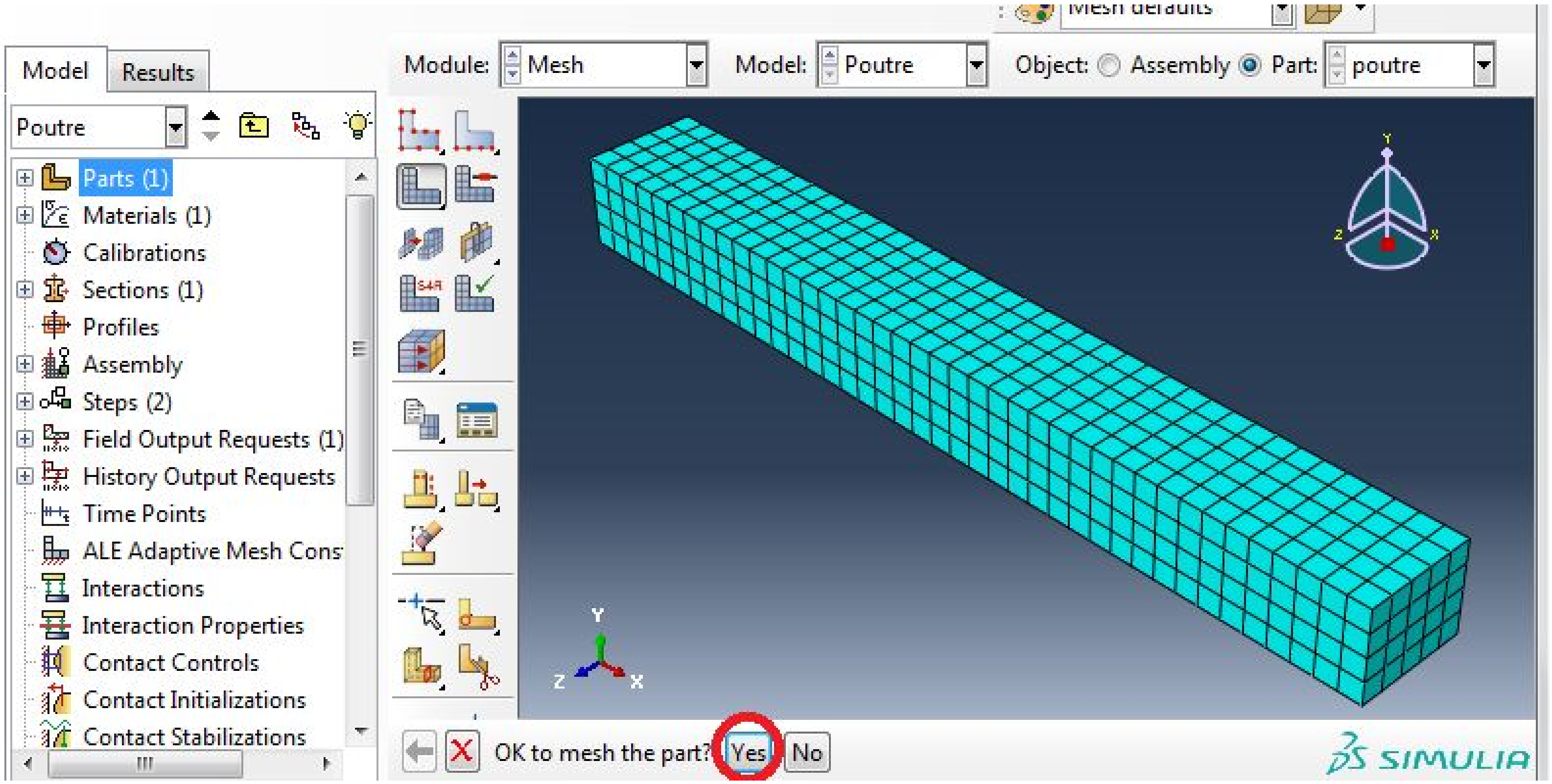
Distortion control: [✓ Use default] Yes No

Length ratio: 0.1

C3D8I: An 8-node linear brick, incompatible modes.

Creation of the Abaqus model

3- From the main menu bar, select **Mesh** → **Part** to mesh the part.



Module: Mesh Model: Poutre Object: Assembly [✓ Part]: poutre

OK to mesh the part? [✓ Yes] No

supméca

Digital Product Simulation

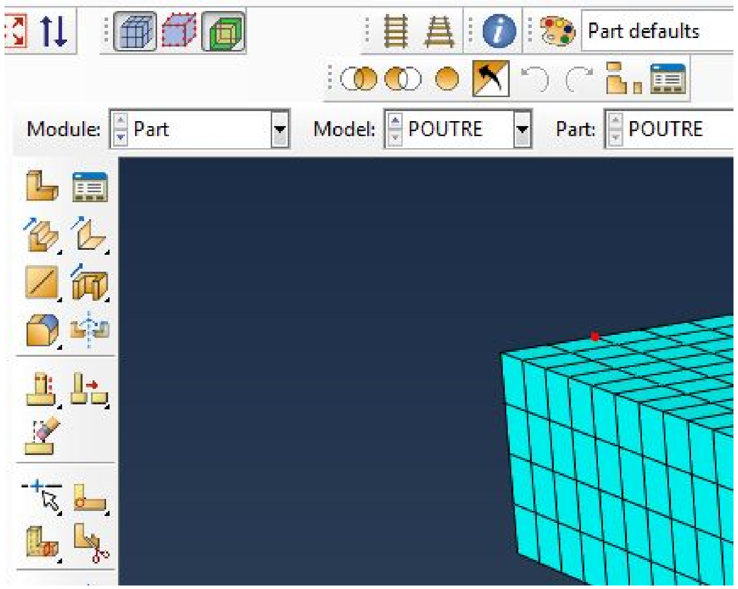
Preparing the Dymola model

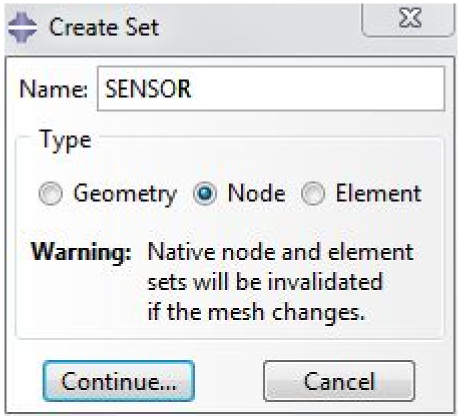
Preparing the Dymola model

1- On the tree under POUTRE part double click on **set**.

2- Name the set **SENSOR** and choose **Node** as a type the click on continue.

3- Select the node on the bottom of the unfixed surface.





Name: SENSOR

Type

Geometry [✓ Node] Element

Warning: Native node and element sets will be invalidated if the mesh changes.

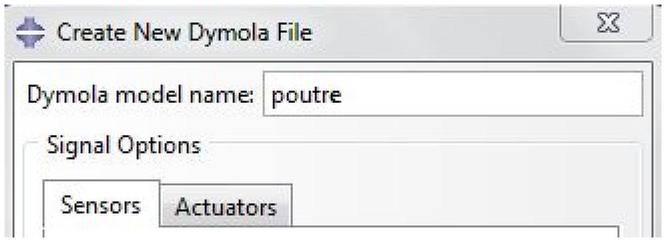
Preparing the Dymola model

4- Go to **job** module on Abaqus.

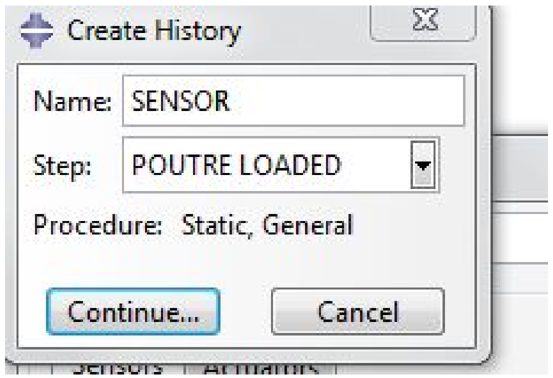
5- Select **plugins** → **Dymola** → **write a file**.

We will create sensors and actuators and create automatically the Dymola file from Abaqus.

6- Name the **Dymola Model** as **poutre**, then Select Create sensor and name it **SENSOR** choose for step the **POUTRE LOADED**


Create New Dymola File

Dymola model name: poutre


Create History

Name: SENSOR

Step: POUTRE LOADED

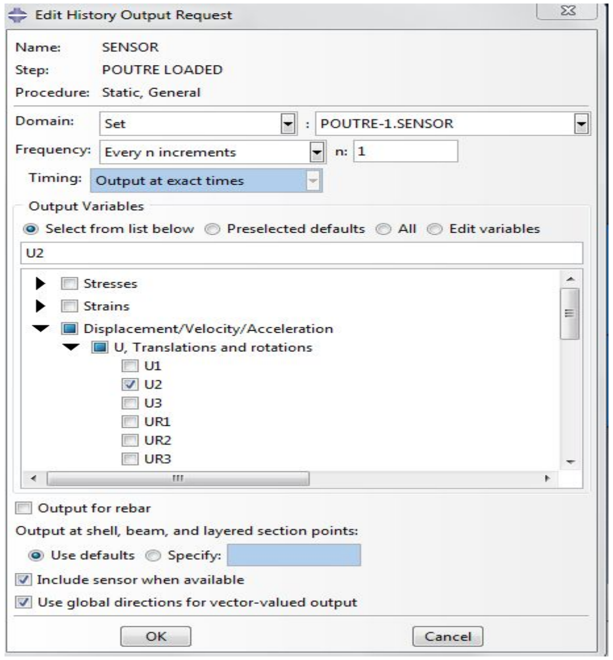
Procedure: Static, General

Preparing the Dymola model

7- On domain choose **set** and choose the **node SENSOR** created previously.

8- Choose the output displacement **U2**.

9- And choose the option **include sensor when available**.



Edit History Output Request

Name: SENSOR
Step: POUTRE LOADED
Procedure: Static, General

Domain: Set : POUTRE-1.SENSOR
Frequency: Every n increments n: 1
Timing: Output at exact times

Output Variables
[✓ Select from list below] Preselected defaults All Edit variables
U2

Stresses
Strains
Displacement/Velocity/Acceleration
U, Translations and rotations
U1
✓ U2
U3
UR1
UR2
UR3

Output for rebar
Output at shell, beam, and layered section points:
[✓ Use defaults] Specify:
✓ Include sensor when available
✓ Use global directions for vector-valued output

OK Cancel

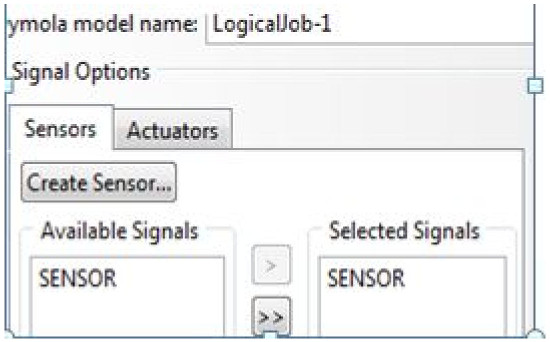
Preparing the Dymola model

10- Click OK.

11- Click add the SENSOR.

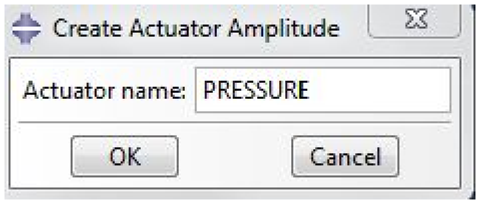
12- Switch now to the Actuators tab.

13- Click on create Actuator and name it **PRESSURE** and click OK.



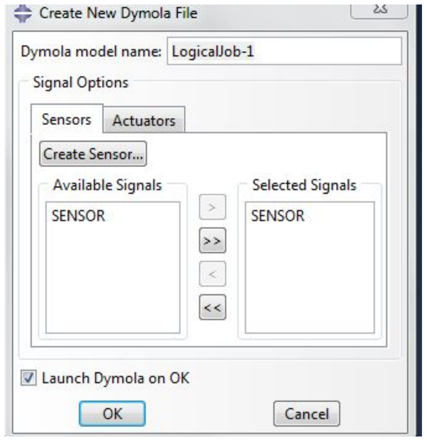
ymola model name: LogicalJob-1

Signal Options



Create Actuator Amplitude

Actuator name: PRESSURE



Create New Dymola File

Dymola model name: LogicalJob-1

Signal Options

☑ Launch Dymola on OK

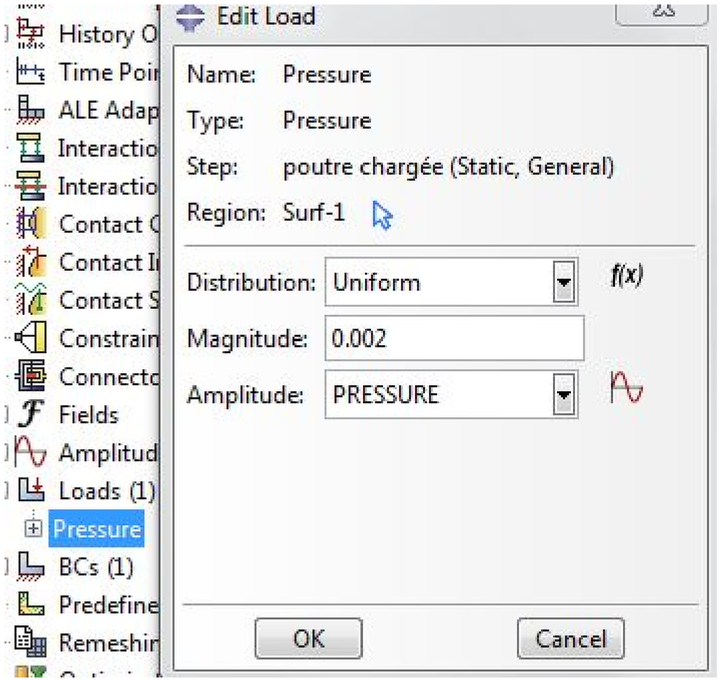
Preparing the Dymola model

14- Add it to the selected signals and then click OK.

15- Dymola will be launched automatically.

16- On abaqus reopen the pressure on the load container and change in **Amplitude** the rampe to the pressure

as shown:



Name: Pressure
Type: Pressure
Step: poutre chargée (Static, General)
Region: Surf-1
Distribution: Uniform
Magnitude: 0.002
Amplitude: PRESSURE

Digital Product Simulation

SETUP DYMOLA

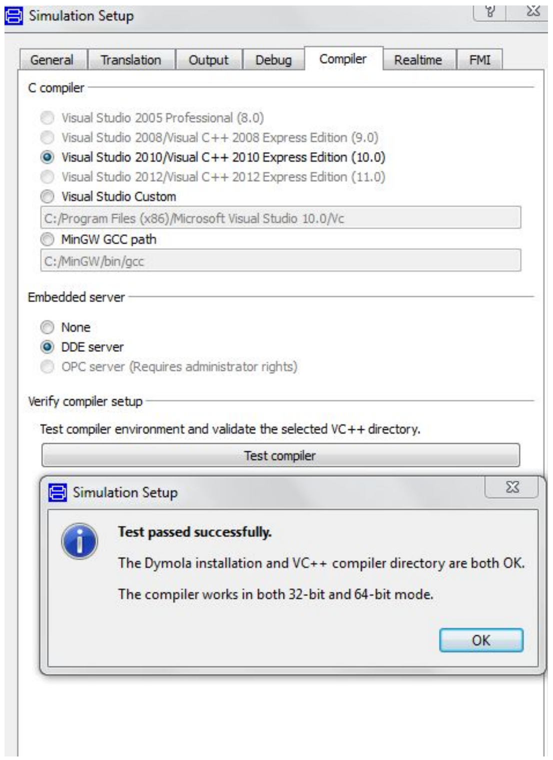
SETUP DYMOLA

1- Click Simulation to switch to the Simulation panel. Select **Simulation** → **Setup** and then click the **Compiler** tab.

2- From the simulation panel, select:

*Visual studio 2010

*DDE server for embedded server.



C compiler

- Visual Studio 2005 Professional (8.0)
- Visual Studio 2008/Visual C++ 2008 Express Edition (9.0)
- [✓ Visual Studio 2010/Visual C++ 2010 Express Edition (10.0)]
- Visual Studio 2012/Visual C++ 2012 Express Edition (11.0)
- Visual Studio Custom

C:/Program Files (x86)/Microsoft Visual Studio 10.0/Vc

- MinGW GCC path

C:/MinGW/bin/gcc

Embedded server

- None
- [✓ DDE server]
- OPC server (Requires administrator rights)

Verify compiler setup

Test compiler environment and validate the selected VC++ directory.

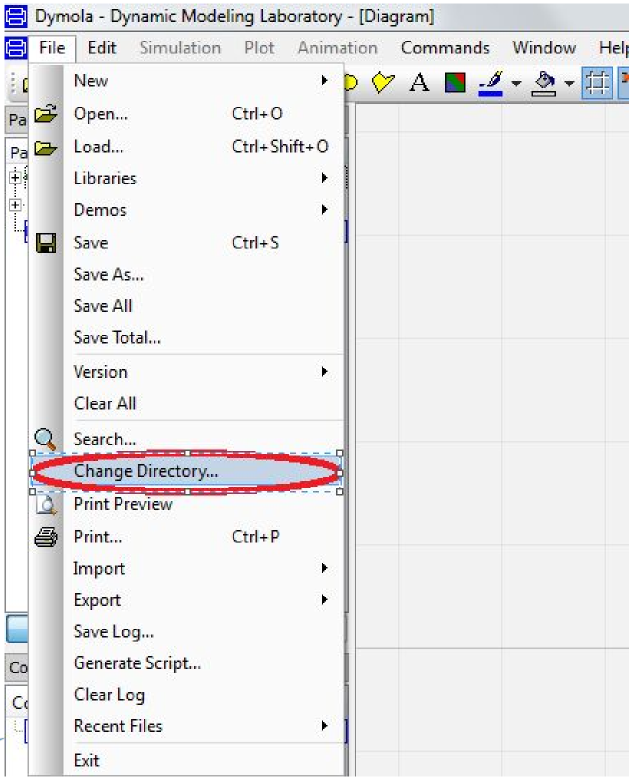
Test passed successfully.

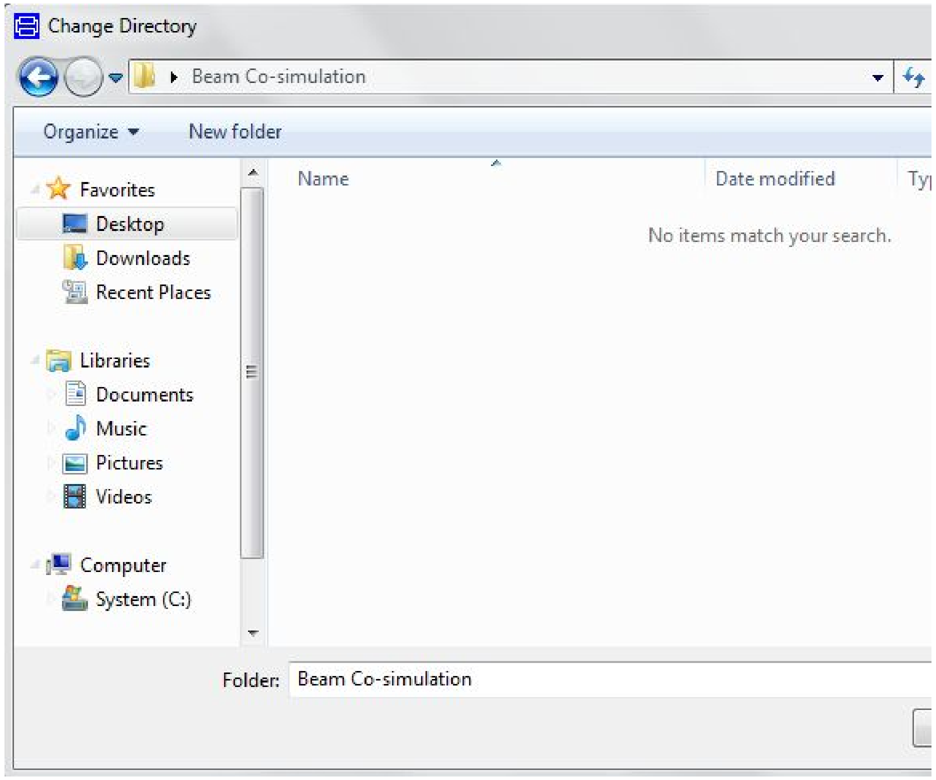
The Dymola installation and VC++ compiler directory are both OK.

The compiler works in both 32-bit and 64-bit mode.

SETUP DYMOLA

3- Verify that the **work directory** for Dymola is the “**Beam co-simulation**”. If it’s not change the directory to this file.





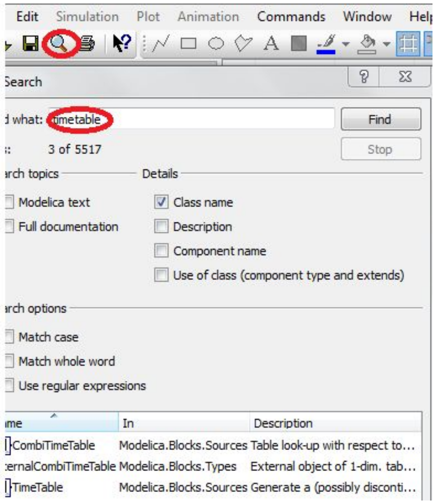
Folder: Beam Co-simulation

SETUP DYMOLA

4- Add and connect these element as shown in the picture:

Click (Bar Menu=>Search):

- Timetable
- Add
- Pi



Search

d what: timetable — Find

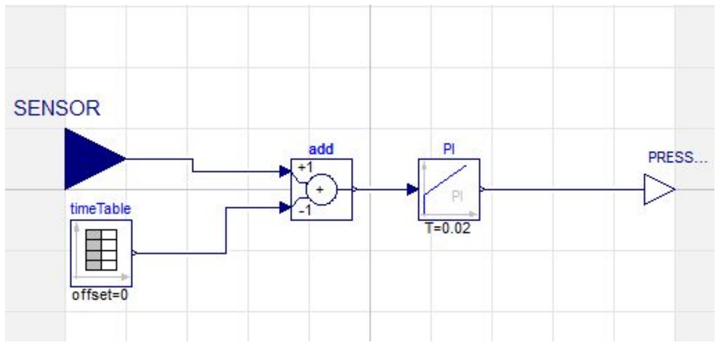
: 3 of 5517 — Stop

rch topics
- [ ] Modelica text
- [ ] Full documentation

Details
- [✓] Class name
- [ ] Description
- [ ] Component name
- [ ] Use of class (component type and extends)

rch options
- [ ] Match case
- [ ] Match whole word
- [ ] Use regular expressions

ime	In	Description
CombiTimeTable	Modelica.Blocks.Sources	Table look-up with respect to...
ernalCombiTimeTable	Modelica.Blocks.Types	External object of 1-dim. tab...
TimeTable	Modelica.Blocks.Sources	Generate a (possibly disconti...

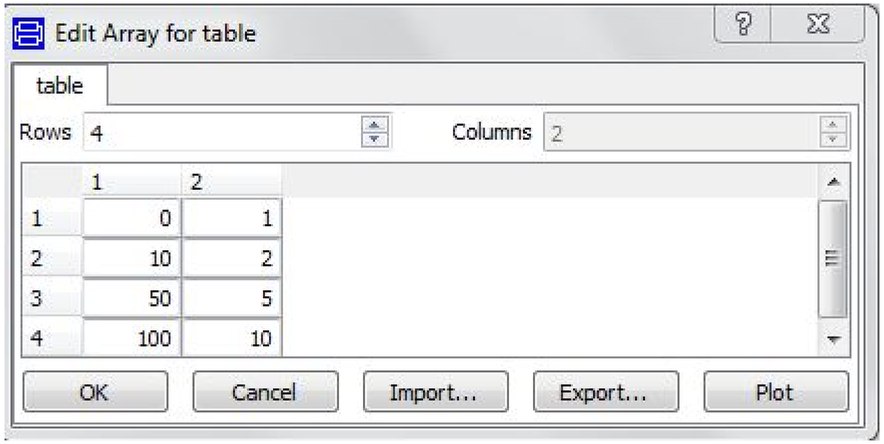


HOW TO SET THE DIFFERENT VALUE OF THE ELEMENT:

The aim of this co-simulation is to compare the position of the node issue from the **sensor** to a target position issued from the **Tabletime** then with the **PI** adjust the pressure to send to Abaqus to reach the Timetable position.

SETUP DYMOLA

5- Insert these values to the **time table (**Displacement in function of time**)**.



Edit Array for table

table

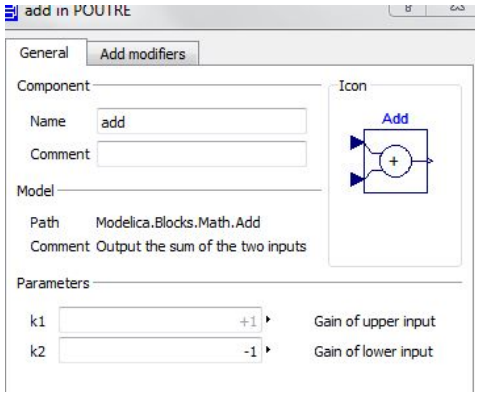
Rows 4 Columns 2

	1	2
1	0	1
2	10	2
3	50	5
4	100	10

time	y

For the **add** insert **k1=+1 ; k2=-1**

And for the **PI** insert **k=0.001 ; T=0.02**



add in POUTRE

Component
Name: add
Comment:

Model
Path: Modelica.Blocks.Math.Add
Comment: Output the sum of the two inputs

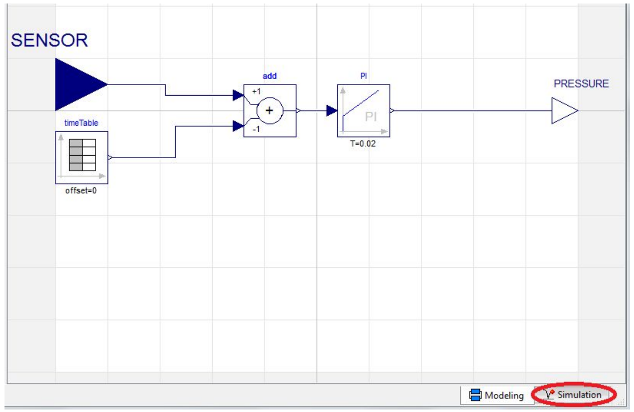
Parameters

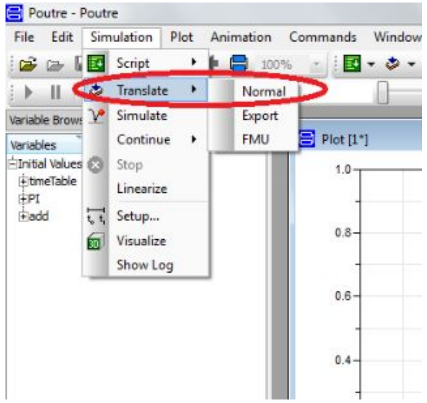
k1	+1	Gain of upper input
k2	-1	Gain of lower input

SETUP DYMOLA

6- Click on Save

Now click on **simulate** and in the Bar tool Click on Simulation=>translate=>Normal





7- **Replace** the dymosim.dll generated on the Dymola **beam co-simulation** by the dymosim generated by industrial license**; if you work with Educative License.**

If you work with **industrial license it will be created automatically.

Launching the co-simulation

Launching the co-simulation

On abaqus:

1- Select **JOB.**

2- Select:**Plugins → Dymola → Submit a Co-simulation.**

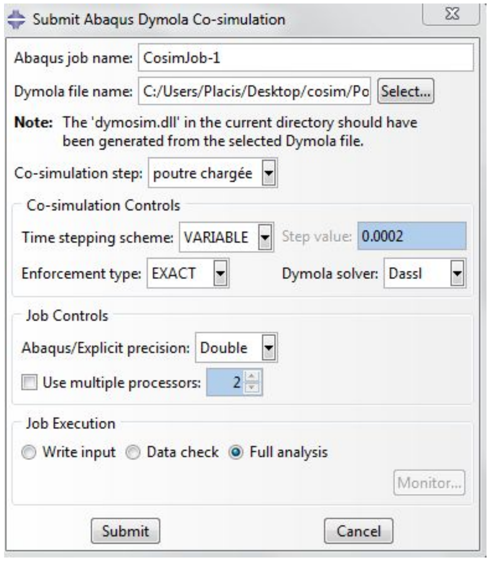
3- In **dymola file name** click on **select** and search for the "**Poutre.mo"** dymola file on the **beam co-simulation** directory.

4- Choose for the step: **POUTRE LOADED**.

5- Choose **variable** for **Time stepping scheme**.

6- Then click **Submit** to start the co-simulation.

7- Click **Monitor** and **refresh**.



Abaqus job name: CosimJob-1

Dymola file name: C:/Users/Placis/Desktop/cosim/Po

Note: The 'dymosim.dll' in the current directory should have been generated from the selected Dymola file.

Co-simulation step: poutre chargée

Co-simulation Controls

Time stepping scheme: VARIABLE Step value: 0.0002

Enforcement type: EXACT Dymola solver: Dassl

Job Controls

Abaqus/Explicit precision: Double

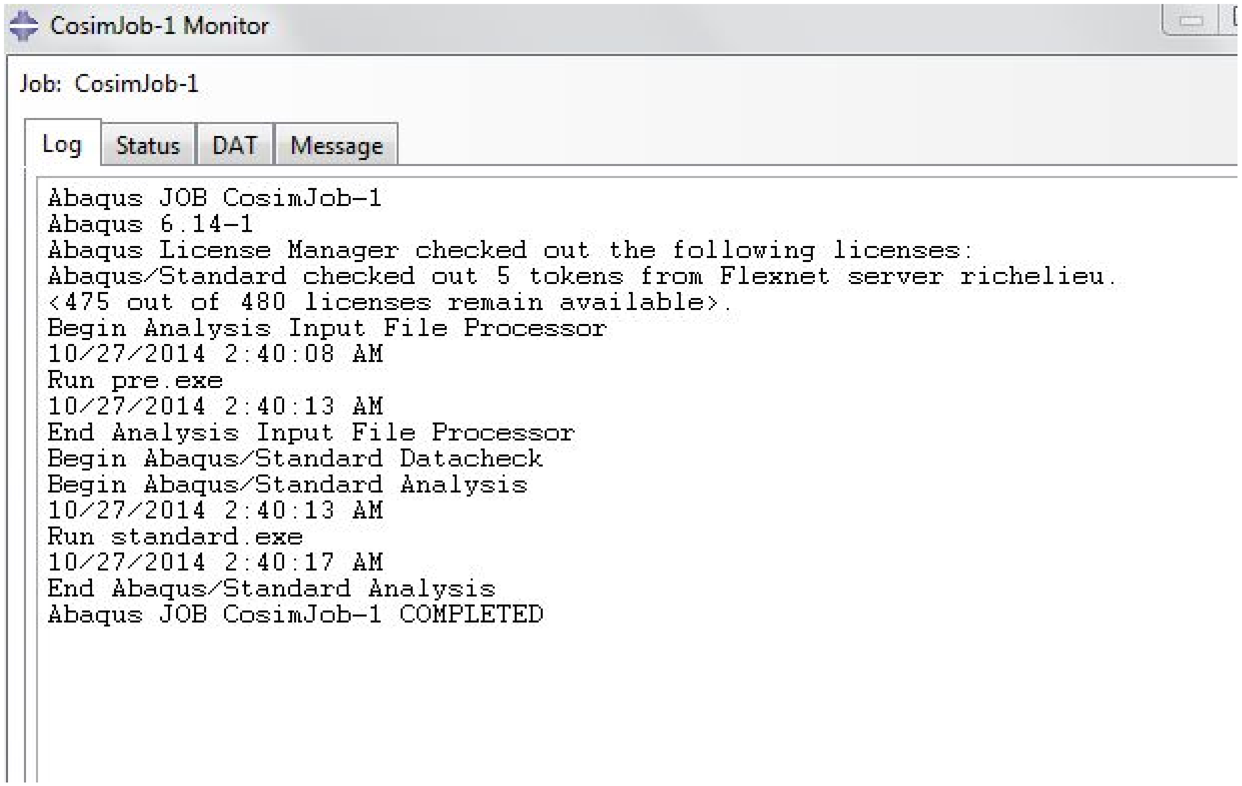
☐ Use multiple processors: 2

Job Execution

○ Write input ○ Data check ● Full analysis

Launching the co-simulation

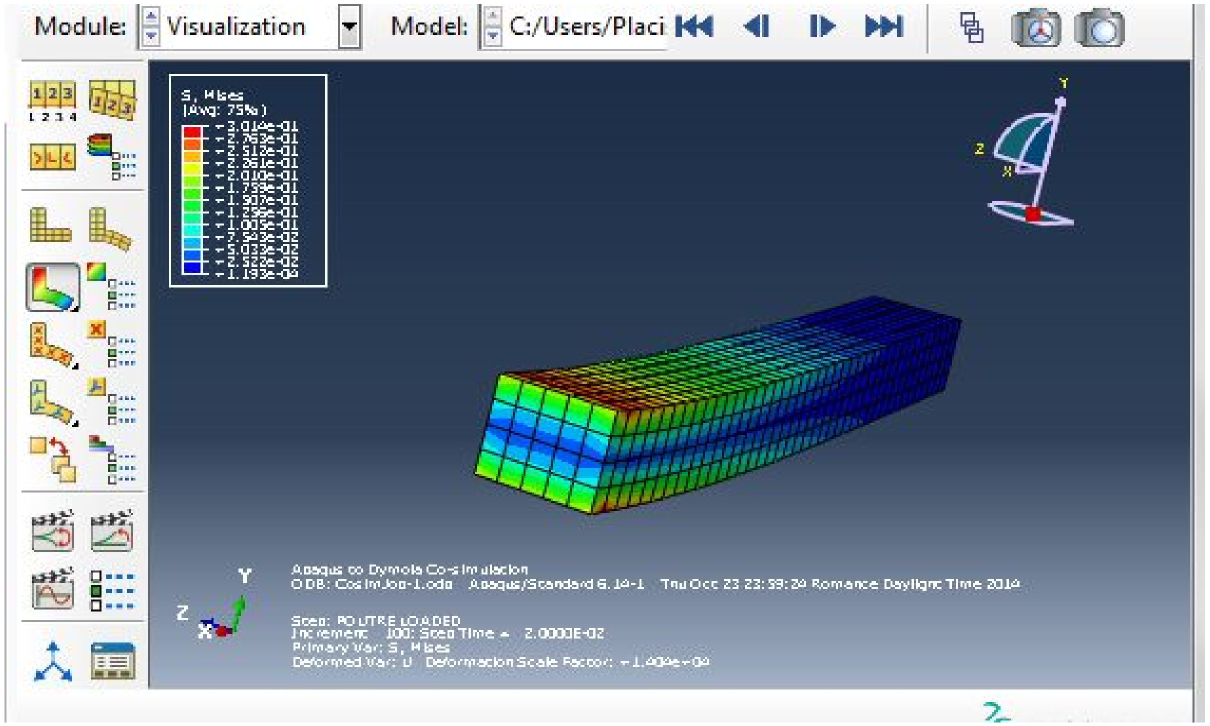
Once it finish open the Abaqus co-simulation job the CosimJob-1.ODB file.



```
Abaqus JOB CosimJob-1
Abaqus 6.14-1
Abaqus License Manager checked out the following licenses:
Abaqus/Standard checked out 5 tokens from Flexnet server richelieu.
<475 out of 480 licenses remain available>.
Begin Analysis Input File Processor
10/27/2014 2:40:08 AM
Run pre.exe
10/27/2014 2:40:13 AM
End Analysis Input File Processor
Begin Abaqus/Standard Datacheck
Begin Abaqus/Standard Analysis
10/27/2014 2:40:13 AM
Run standard.exe
10/27/2014 2:40:17 AM
End Abaqus/Standard Analysis
Abaqus JOB CosimJob-1 COMPLETED
```

Launching the co-simulation

Result:



S, Mises
(Avg: 75%)

```
+3.014e-01
+2.763e-01
+2.512e-01
+2.261e-01
+2.010e-01
+1.759e-01
+1.507e-01
+1.256e-01
+1.005e-01
+7.543e-02
+5.033e-02
+2.522e-02
+1.193e-04
```

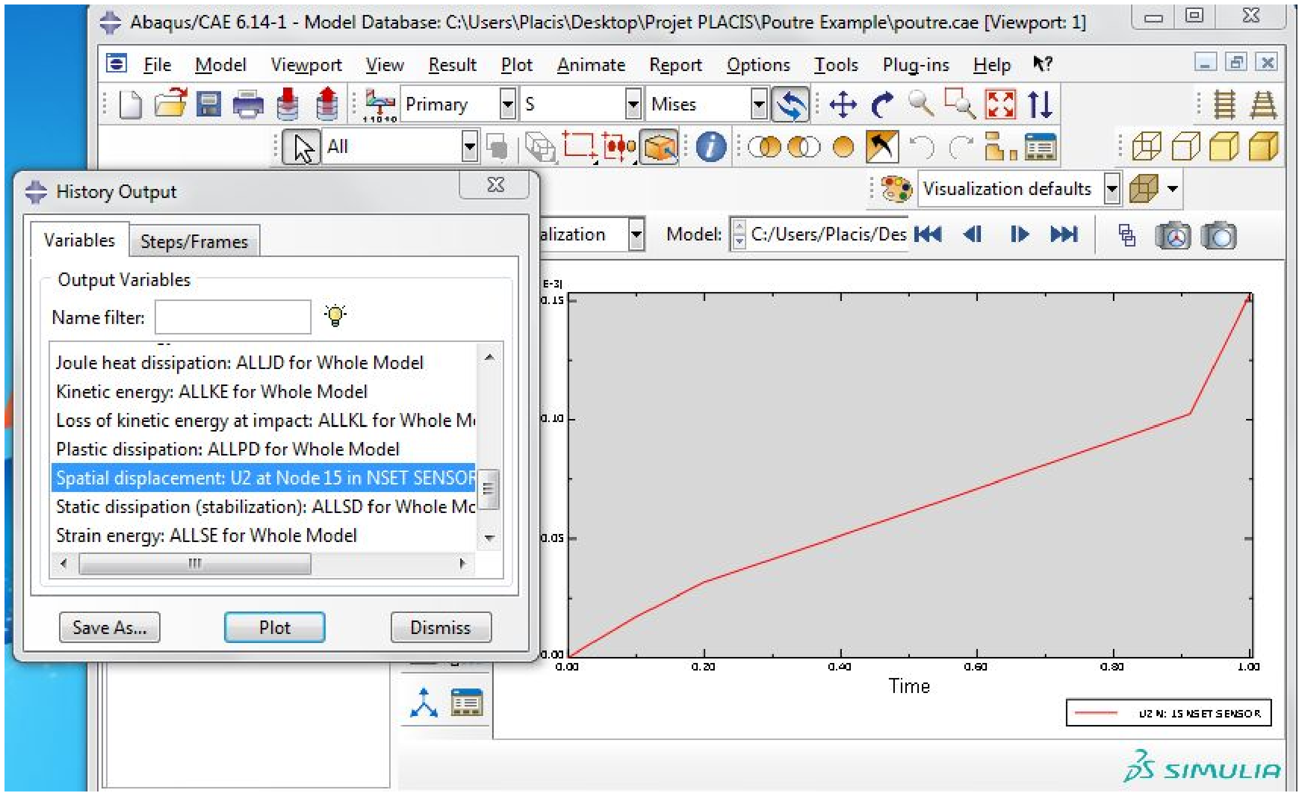
```
Abaqus to Dymola Co-simulation
ODB: CosimJob-1.odb    Abaqus/Standard 6.14-1    Thu Oct 23 23:59:24 Romance Daylight Time 2014

Step: POUTRE LOADED
Increment    100: Step Time =    2.0000E-02
Primary Var: S, Mises
Deformed Var: U   Deformation Scale Factor: +1.404e+04
```

Launching the co-simulation

In the menu bar click **Result** → **History Output**.

Chose *Spatial displacement U2* and click Plot.



Joule heat dissipation: ALLJD for Whole Model
Kinetic energy: ALLKE for Whole Model
Loss of kinetic energy at impact: ALLKL for Whole M
Plastic dissipation: ALLPD for Whole Model
Spatial displacement: U2 at Node 15 in NSET SENSOR
Static dissipation (stabilization): ALLSD for Whole Mo
Strain energy: ALLSE for Whole Model

Cosimulation: Graphical method

(with beam file already done)

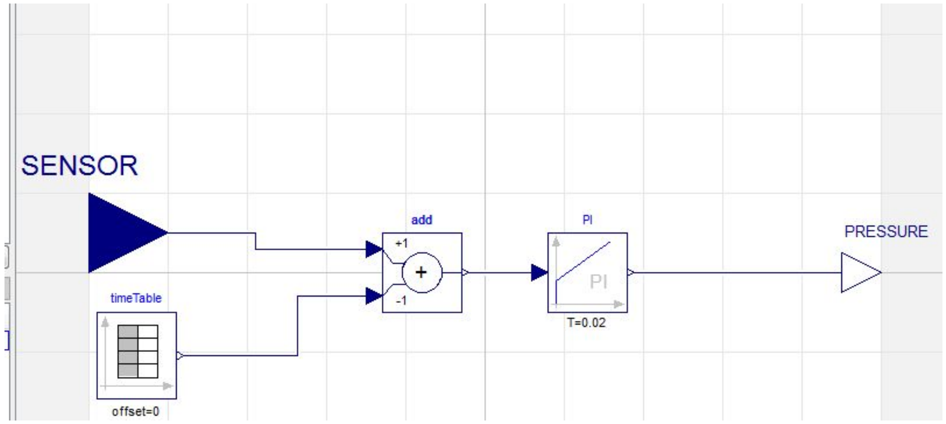
Digital Product Simulation

Cosimulation: Graphical method

1- Launch Dymola and open the model poutre.mo

2- Regroupe this files in one file and name it **Beam_Cosim**

- Poutre.cae
- Poutre.INP
- Poutre.mo
- Poutre.JNL
- Libsdll.DLL



Cosimulation: Graphical method

From the bottom of the Modeling panel in Dymola,

3- Click Simulation to switch to the Simulation panel. Select **Simulation** → **Setup** and then click the **Compiler** tab.

Compiler à Visual Studio 2010/Visual C++ 2010 Express Edition(10.0).

4- Select **Simulation** → **Translate**. In your current working directory you should see a file called **dymosim.dll** that was just created by the translation.

5- Copy the Dymosim.dll in **Beam_cosim** file (if you work with Educative license).

6- Import the **INP** file (Poutre.INP).

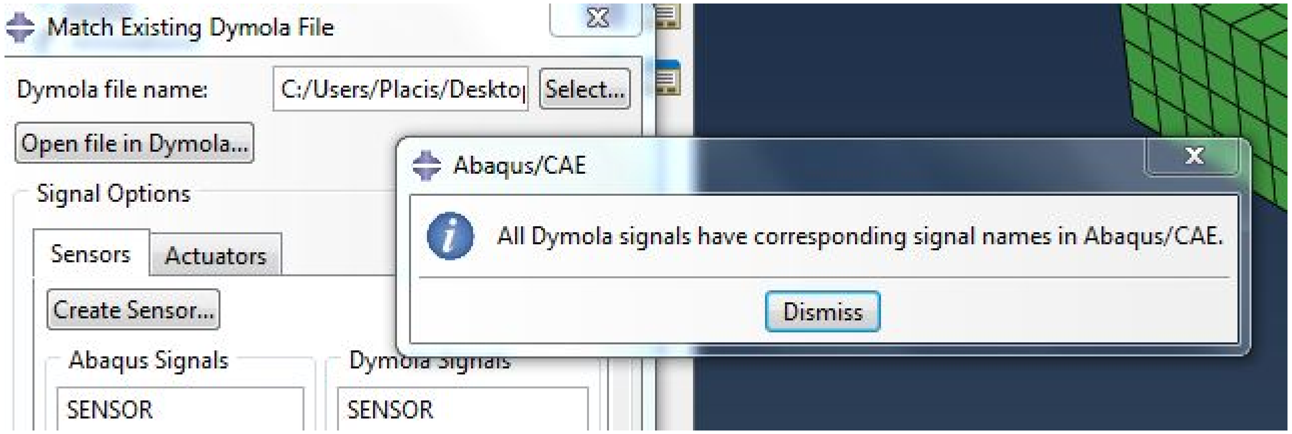
7- Set **Beam_cosim** as the work directory of Abaqus.

.

Cosimulation: Graphical method

8- Select **Job → Plug_in → Dymola → Match files**

Dymola file name: Select the file of the Dymola logical (*.mo) that will be read into the plug-in


Match Existing Dymola File

Dymola file name: C:/Users/Placis/Deskto|

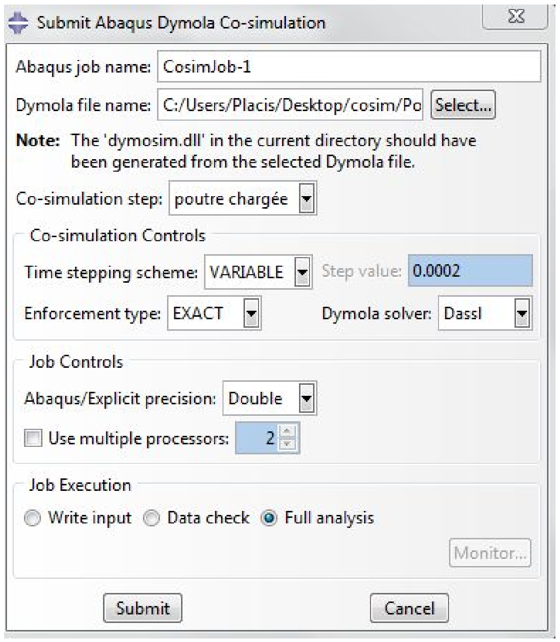
Abaqus/CAE

All Dymola signals have corresponding signal names in Abaqus/CAE.

Cosimulation: Graphical method

9- Select: **Plugins → Dymola → Submit a Co-simulation.**

- In **dymola file name** click on **select** and choose for the “**Poutre.mo”** dymola file on the **beam co-simulation** directory.
- Choose for the step: **POUTRE LOADED**.
- Choose **variable** for **Time stepping scheme**.
- Then click **Submit** to start the co-simulation.
- Click **Monitor** and **refresh**.
- Once it finish open the Abaqus co-simulation job the CosimJob-1.ODB file.



Submit Abaqus Dymola Co-simulation

Abaqus job name: CosimJob-1

Dymola file name: C:/Users/Placis/Desktop/cosim/Po

Note: The 'dymosim.dll' in the current directory should have been generated from the selected Dymola file.

Co-simulation step: poutre chargée

Co-simulation Controls

Time stepping scheme: VARIABLE — Step value: 0.0002

Enforcement type: EXACT — Dymola solver: Dassl

Job Controls

Abaqus/Explicit precision: Double

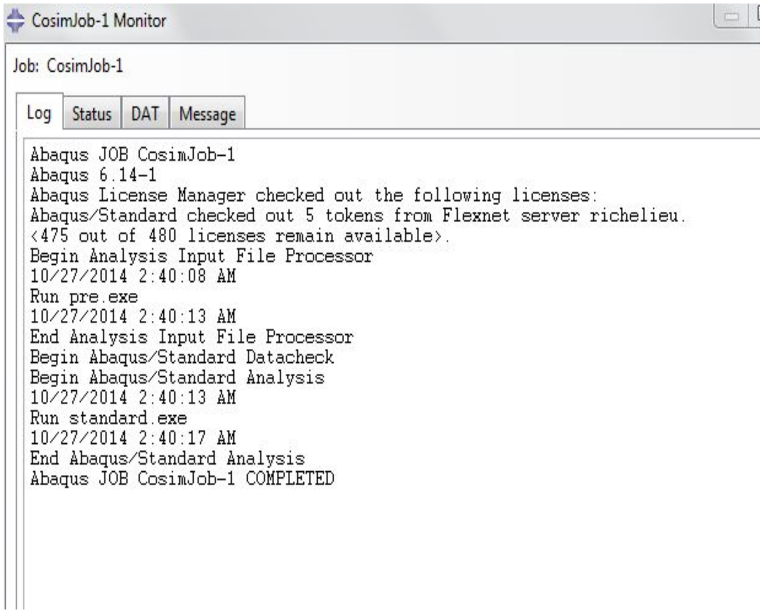
Use multiple processors: 2

Job Execution

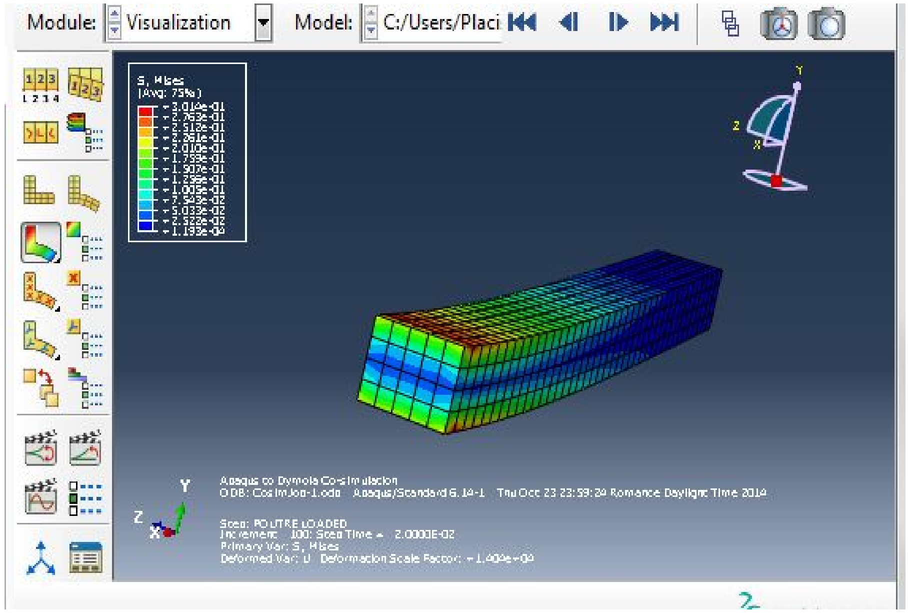
Write input — Data check — [✓ Full analysis]

Cosimulation: Graphical method

- Once it finish open the Abaqus co-simulation job the **CosimJob-1.ODB** file



```
Abaqus JOB CosimJob-1
Abaqus 6.14-1
Abaqus License Manager checked out the following licenses:
Abaqus/Standard checked out 5 tokens from Flexnet server richelieu.
<475 out of 480 licenses remain available>.
Begin Analysis Input File Processor
10/27/2014 2:40:08 AM
Run pre.exe
10/27/2014 2:40:13 AM
End Analysis Input File Processor
Begin Abaqus/Standard Datacheck
Begin Abaqus/Standard Analysis
10/27/2014 2:40:13 AM
Run standard.exe
10/27/2014 2:40:17 AM
End Abaqus/Standard Analysis
Abaqus JOB CosimJob-1 COMPLETED
```



```
Abaqus to Dymola Co-simulation
ODB: CosimJob-1.odb    Abaqus/Standard 6.14-1    Thu Oct 23 23:59:24 Romance Daylight Time 2014

Step: POUTRE LOADED
Increment    100: Step Time =    2.0000E-02
Primary Var: S, Mises
Deformed Var: U   Deformation Scale Factor: +1.404e+04
```

Digital Product Simulation

THANK YOU FOR YOUR ATTENTION!

supméca

PLACIS

Digital Product Simulation