

EXERCISE N.2 – ENGINEERING CALCULATIONS & FEM

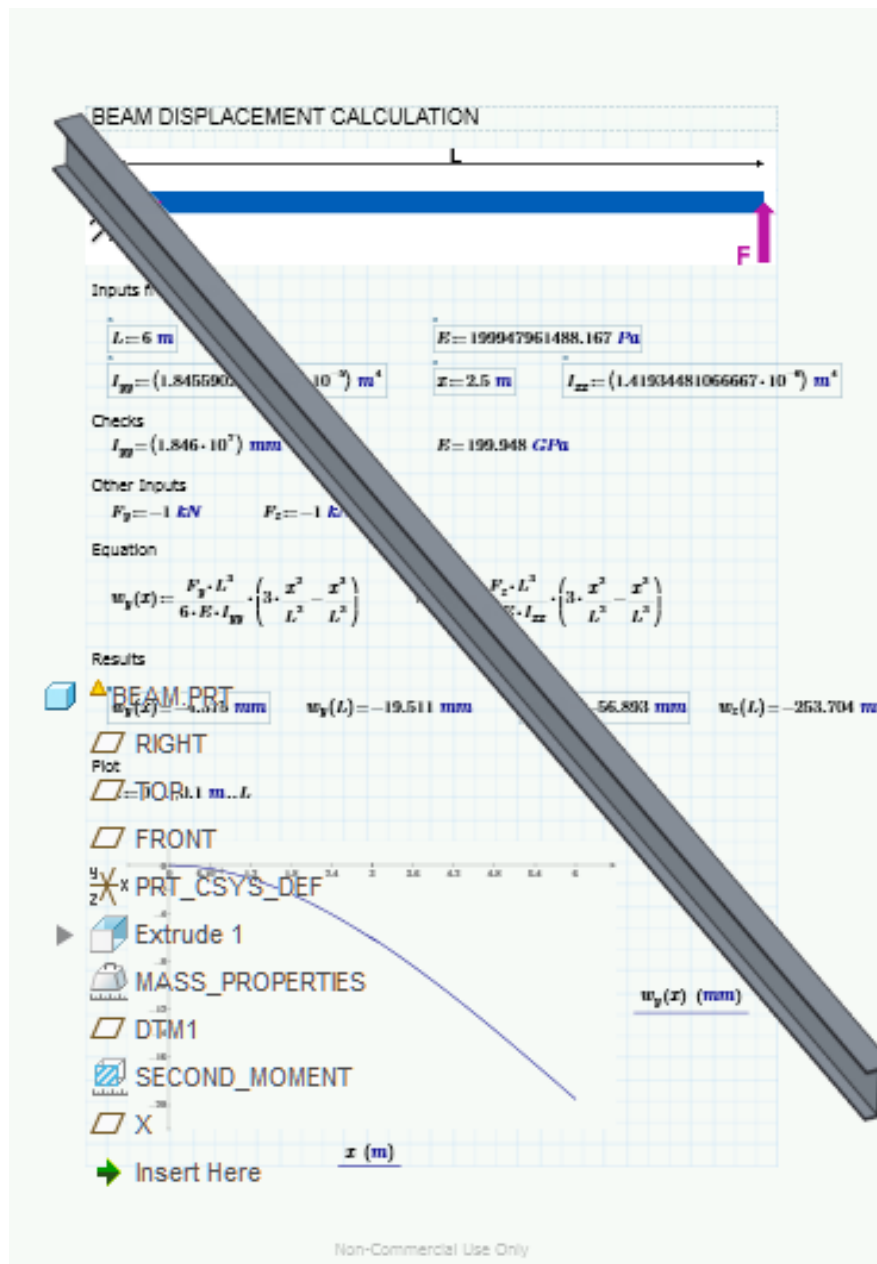


Figure 1: Model used in this exercise and Mathcad worksheet.

Learning Targets

In this exercise you will learn

- ✓ to use palette in sketching
- ✓ to make cross-section and mass analysis
- ✓ to create engineering calculations using Mathcad
- ✓ to create a basic FEM analyze using Creo Simulate.

This exercise shows how geometry information from Creo 3.0 can be used in engineering calculations (Mathcad Prime 3.1) and how to perform a simple strength analysis using Creo.

Sometimes it is necessary to make engineering calculations in a separate application, for ex. if needed more complicate operations than those existing in Creo (integration and derivation, graphs, etc.). In this exercise, a strength analysis of an I-beam will be created. Creo creates geometry and Mathcad solves the equations and graphs results. Finally, a FEM model will be created and calculations validated using Creo Simulate.

Getting Started

Create a **new** solid model and name it as BEAM.

Geometry

Extrude

Create an **Extrude** (📐) to the RIGHT plane using TOP as a reference plane (*Orientation Top*) and go to sketch mode. Select **Palette** (🌀) from *Sketching* group. A *Sketcher Palette* window appears. From this window, different kind of predefined shapes can be found. Select **Profiles** tab. Select **I-profile** and drag & drop it on the sketching area (Figure 2).

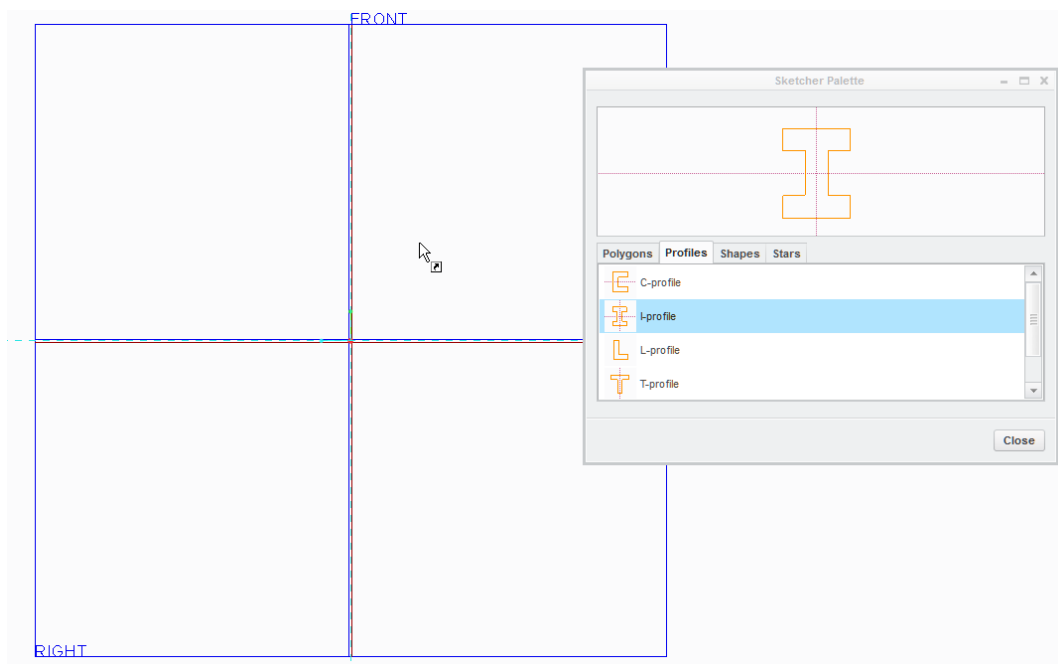


Figure 2: I-profile selected and draping it to the sketching area.

Notice that this I-profile has an X symbol in the middle. Drag the profile from that symbol downwards, until it snaps to the horizontal reference line (Figure 3). Drag it a little bit more to snap it to the vertical reference line. When placed in the middle, accept (✓) the *Import Section*. **Close** the *Sketcher Palette* window.

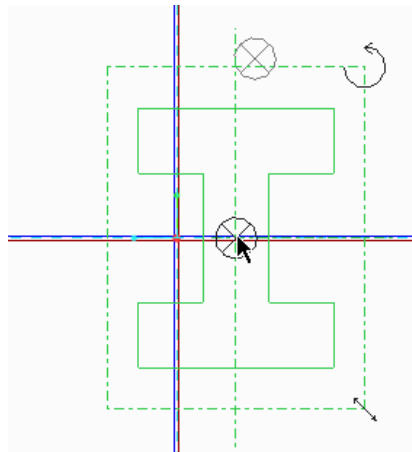


Figure 3: Profile snapped to the horizontal reference line.

Update the dimension values as seen in Figure 4. (IPE type 200 mm I-profile)

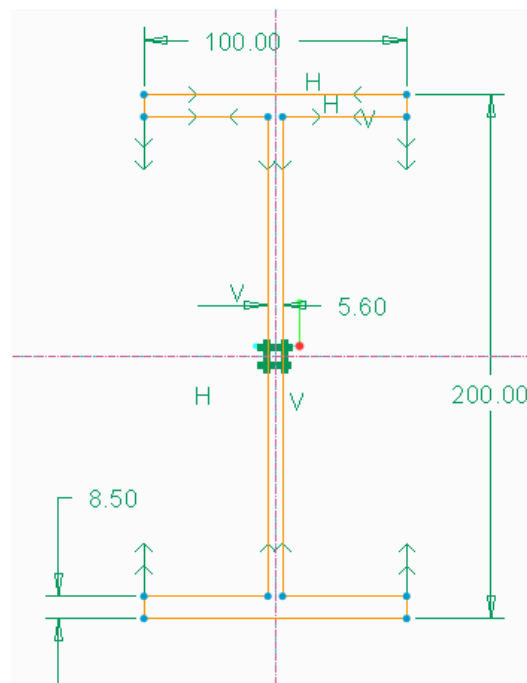


Figure 4: Finished sketch.

When ready, accept the sketch. Extrude it **6000**.

Plane

Create a **Plane** (□) **3000** offset from RIGHT plane.

Measurements

Select **Analysis** tab. Select **X-Section Mass Properties** (📐) from the *Model Report* group (Figure 5).

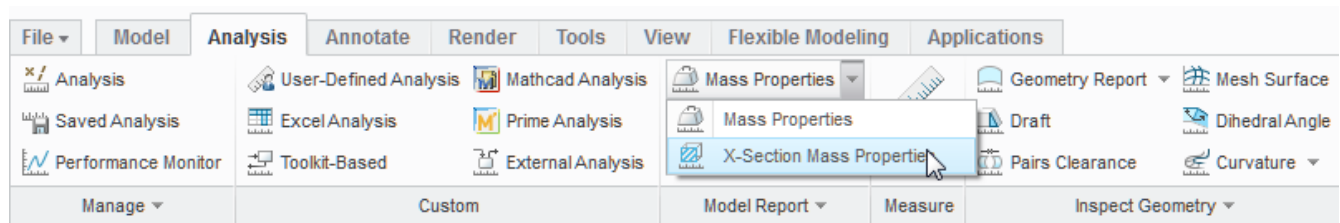


Figure 5: Selecting *X-Section Mass Properties*.

Select the previously created plane as *Plane*. The values updates and you can see different kind of measurement related to this cross section. Replace *Quick* with **Feature** from the drop-down menu. Then you can select the **Feature** tab. Here we can define witch values we will save to the feature. Uncheck XCEC_AREA, check **XSEC_IXX** and **XSEC_IYY**. This two are second moments of areas, needed for ex. in strength calculations. If you zoom a little bit to the cross-section, you can see coordinate system used in this analysis (x sideward, y upward). Save analysis as **Second_Moment** and select **OK**.

Create also **Mass Properties** (📐) calculation from *Model Report* group (Figure 6).

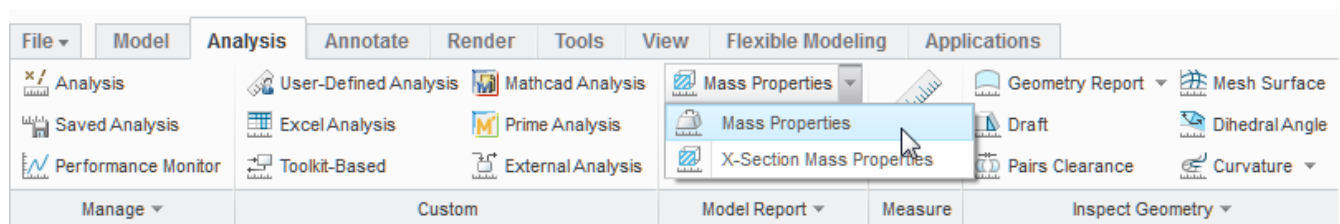


Figure 6: Selecting *Mass Properties*.

Change type from *Quick* to **Feature** and select **Preview**. Here you can see mass properties of the whole part, for ex. its mass (about 128 kg). Then go to **Feature** tab. Notice that volume, surface area and mass are already checked. Check also **PNT_COG** from *Datums* list to create a center of mass point in the model. Rename feature as **Mass_Properties** and accept it (**OK**).

Redefining Datum Plane

Next we link the previously created datum plane (DTM1) to the center of the mass point. Why? The X-cross section analysis works only if we have a cross section to analyze, and this way we ensure, that if the length of the beam decreases, the analysis still works. You can test this by changing the offset value of DTM1 to something bigger than 6000 – the analysis will fail. Change it back to 3000.

Select **MASS_PROPERTIES** from the *Model Tree* and move it before DTM1 (Figure 7).

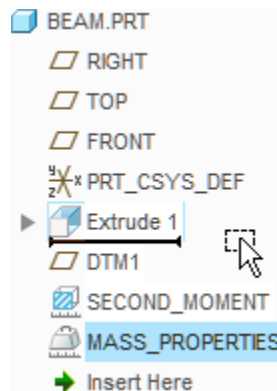


Figure 7: Moving MASS_PROPERTIES before DTM1.

Select **DTM1** from model tree and choose **Edit Definition** (🔧). Hold **CTRL** and select **PNT_COG** as a new reference to our plane (Figure 8, **Point Display** (☒) needs to be on). Notice, that *Offset* is gone. Accept the feature (**OK**).

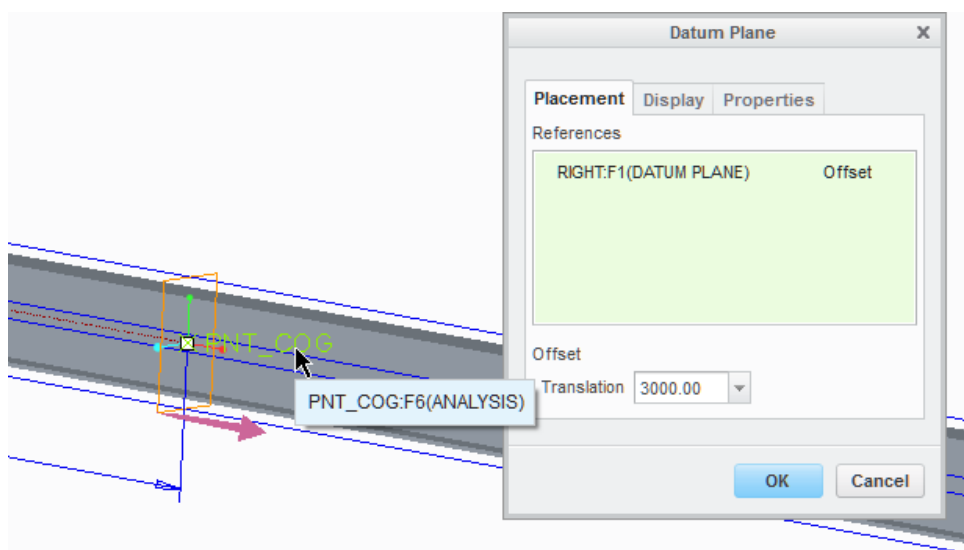


Figure 8: Holding CTRL and selecting PNT_COG.

Mathcad Integration

Next we will integrate Mathcad worksheet to our Creo model. This option works with Creo 3.0 M030 and newer when using Mathcad Prime 3.1 or newer.

Select **Applications** tab and select **Open/Create Worksheet** (M).

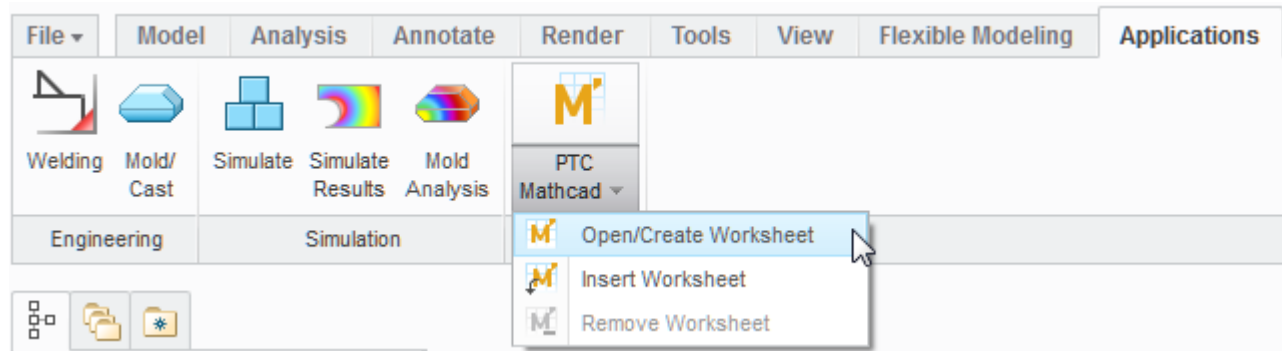


Figure 9: Selecting *Open/Create Worksheet*.

Mathcad session will open (this may take some time when using first time). Here we can create our functions, plots and reports, and this information will be embedded to the Creo file (no separate Mathcad file needed). As you can see, also Mathcad uses Ribbon interface.

Inputs

First we need to define what we will import from Creo model. Select **Document** tab and select **Text Block** (A) from *Regions* group. Write *Inputs from Creo* to the field. It is a good practice to comment calculations, it makes easier to change them later. Click somewhere below previously created line (Figure 10).

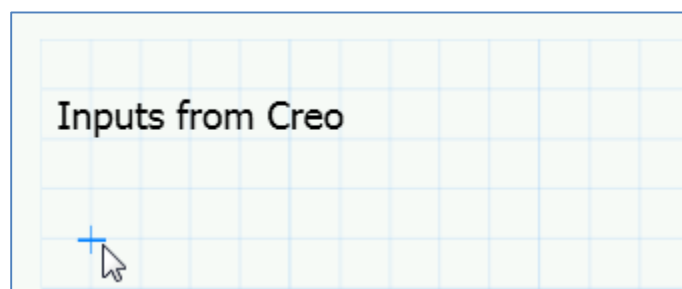


Figure 10: Text block created, clicked below.

About equation formation in Mathcad

Mathcad uses two different kind of equal (=) definitions:

- Normal equal (=) is to show the value of an existing variable.
- Definition equal (:=) is to define a variable and give a value to it.

Also, Mathcad understand and uses units (mm, kg, °C etc.), so when defining a variable, we can also give it a unit (for ex. 6 m).

Creating Inputs

To define L variable with a value of 6 m (the length of our beam), write.

L	:	6	m	ENTER
---	---	---	---	-------

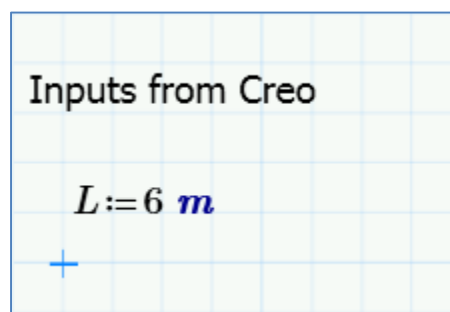


Figure 11: Variable L defined. Notice, that units are highlighted in blue.

Now we have our first variable defined (Figure 11). We need to define three more inputs (second moment of area, elastic modulus, measurement point). Let's define the second moment of area, write

I	CTRL+-	yy	:	1	m^4
---	--------	----	---	---	-----

The CTRL+- command created a subscript to our variable name. You can also create subscripts by selecting **Subscript** (a_2) from *Style* group in the **Math** tab.

Next, create variables as seen in Figure 12.

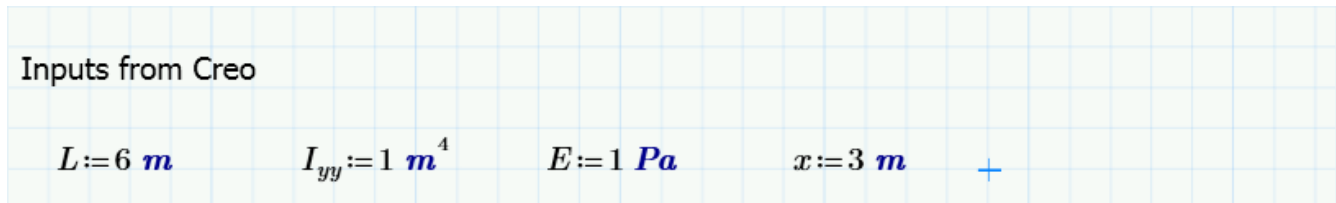



Figure 12: Defined variables.

Create another **Text Block** () from **Math** tab and write *Other Inputs*. Then, create a variable as seen in Figure 13.

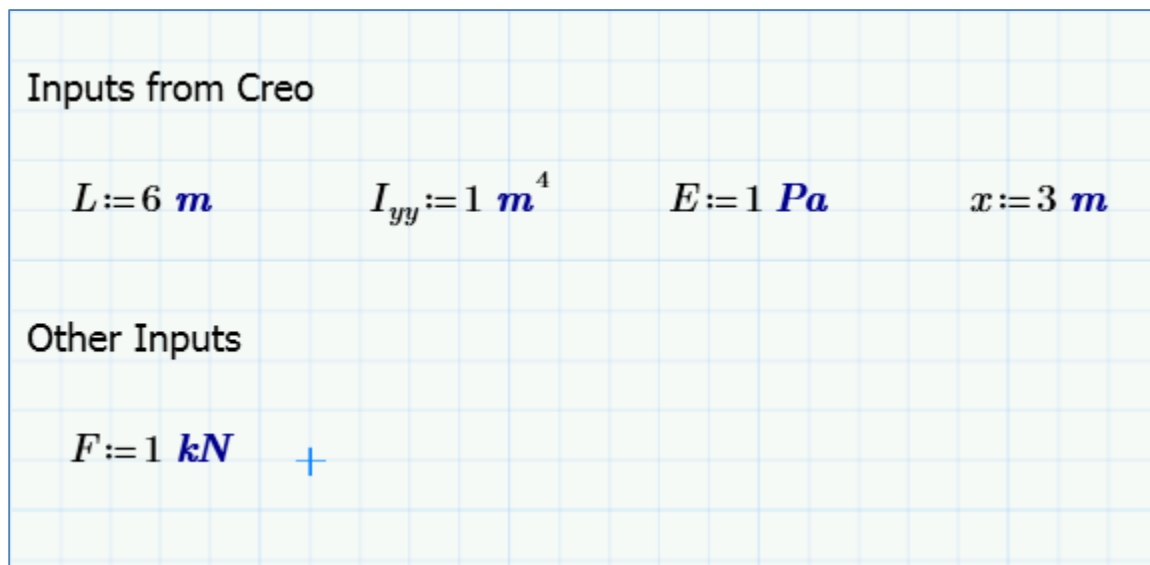



Figure 13: Currents status of the worksheet.

Define as Inputs

We have variables, next we need to define input variables from Creo. Select **Input/Output** tab. Select all four variables above Inputs from Creo text field (**CTRL** to select several). Then select **Assign Inputs** () from Integration field (Figure 14). There should be small *in* text above selected fields to mark input field. Right now we don't need output fields (also, output fields needs to be with normal equal = sign).

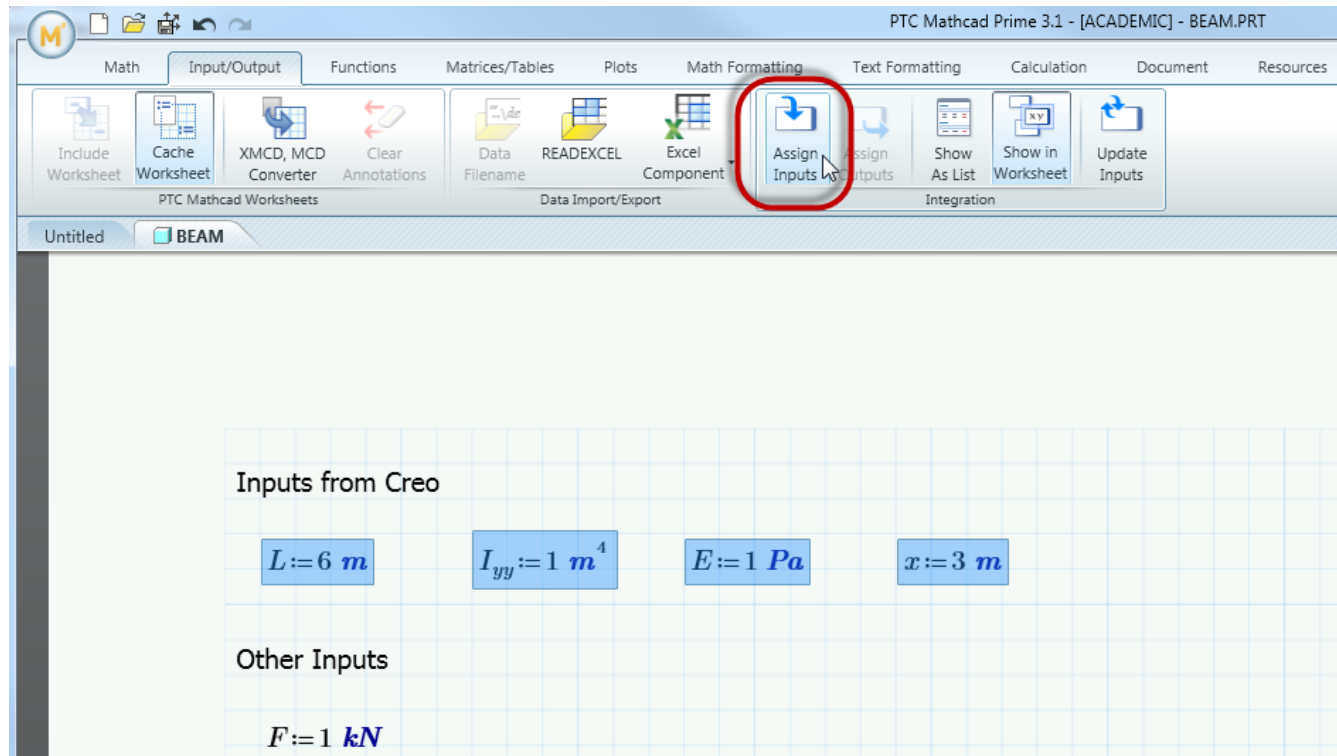


Figure 14: Four variables selected (L, I, E, x) and selecting Assign Inputs.

Updating Creo Model

We need to define the connection between Mathcad document and Creo model. Select **Save and Push** (📁) to push created output variables to Creo model.

Let the Mathcad document to be open in the background. Go to Creo. Create a new **Datum** (📏) to be offset from RIGHT plane and give **4500** as an offset value. Rename plane as X.

Relations

Select **Relations** (d=) from *Model Intent* group. Select **Insert** → **From List**. As *Look In*, select **Embedded Mathcad**, select **IYY** and **Insert Selected** (Figure 15).

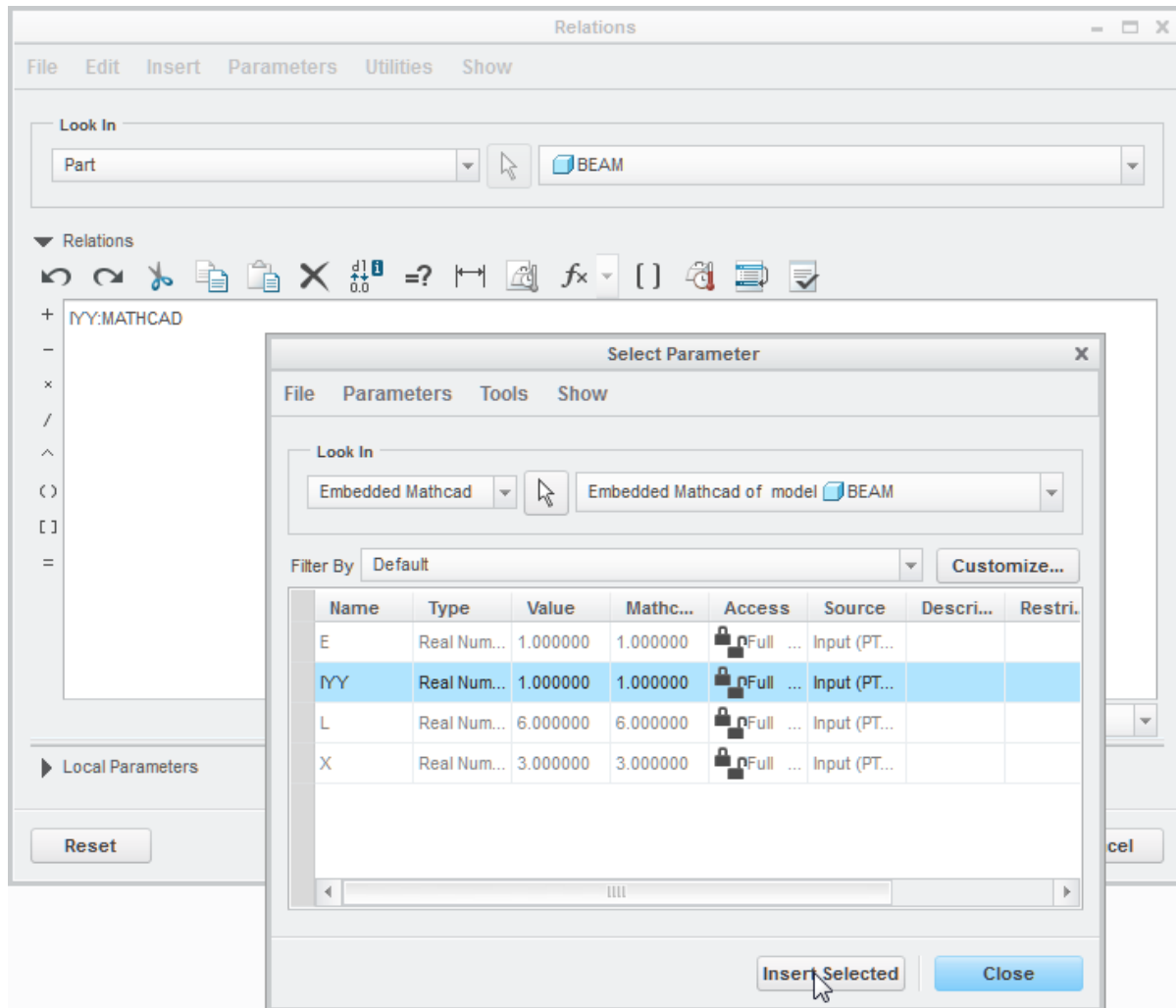


Figure 15: IYY selected, selecting Insert Selected.

Now we have a Mathcad variable in our *Relations* field. Notice the naming scheme of Mathcad parameters. They are always named as [Name]:MATHCAD (subscripts are ignored and added directly to parameter names). Write = after IYY:MATHCAD. Then select **Insert** → **From List** again. As *Look In*, select **Feature**, select **SECOND_MOMENT** from the model tree and add **XSEC_IYY** to the *Relations*. Now there is a link between Mathcad variable and Creo analysis.

Update *Relations* field as seen in Figure 16. Notice, that in picture, FID_280 is SECOND_MOMENT.

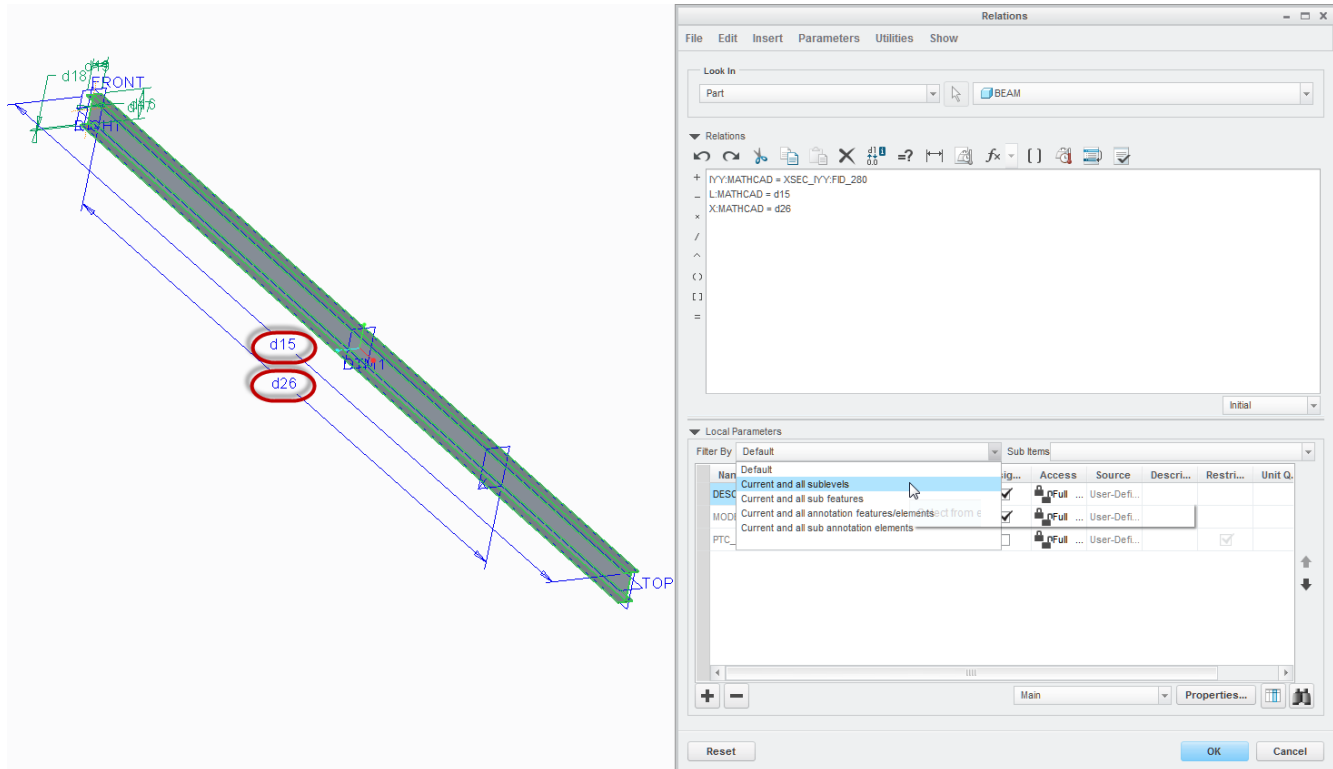


Figure 16: Updated Relations.

We are missing one variable, E (elastic modulus). To see it, select **Local Parameters** and **Current and all sublevels** (Figure 16). Scroll down in the list and select **PTC_YOUNG_MODULUS**, RMB and select **Insert to Relations**. As you can see, also all analysis are listed there. Your relations should look like in Table 1.

Table 1: Relations in BEAM.prt. You may have different names.

IYY:MATHCAD = XSEC_IYY:FID_280 L:MATHCAD = d15 X:MATHCAD = d26 E:MATHCAD = PTC_YOUNG_MODULUS:MTRL_40

Accept Relations (**OK**).

Updating MathCAD Worksheet

Changing units

Select **Input/Output** tab and select **Update Inputs** (🔧) from *Integration* group. The defined input variables are updated (Figure 17). You should a little bit fine-tune locations of the variables to make out Mathcad document clearer.

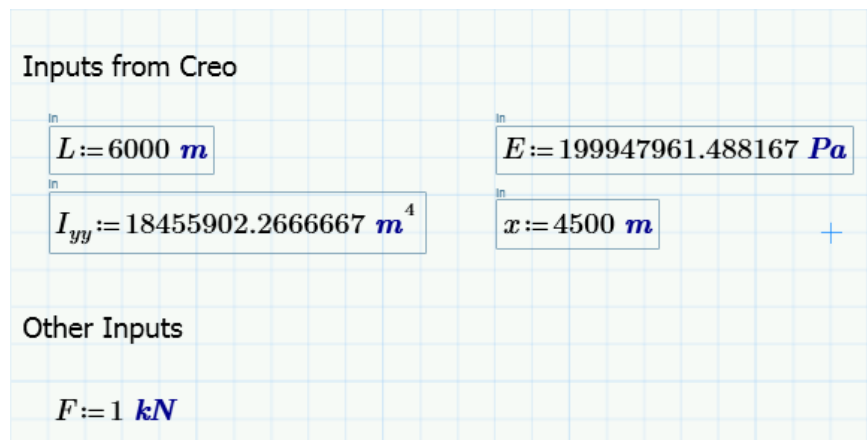


Figure 17: Values in the inputs. Layout updated to see the values (drag & drop).

As you can notice, the values are interesting. Length was 6000 mm, not meters. MathCAD assumes, that all given values are in SI format (meters, kilograms, Pascal etc.), but those are something else in Creo (mm, kg, kPa). Go back to Creo and open **Relations** (d=). Check that *Filter By* is *Current and all sublevels* and look at the *Local Parameters* field (Figure 18).

Owner	Name	Type	Value	Desig...	Access	Source	Descri...	Restri...	Unit Q...	Unit
STEEL	PTC_FATIGUE_TYPE	String	NONE		Locke...	Material		<input checked="" type="checkbox"/>		
STEEL	PTC_MATERIAL_SUB_TYPE	String	LINEAR		Locke...	Material		<input checked="" type="checkbox"/>		
STEEL	PTC_YOUNG_MODULUS	Real Num...	199947961.4...		Full ...	Material		<input checked="" type="checkbox"/>	Stress	kPa
STEEL	PTC_POISSON_RATIO	Real Num...	0.270000		Full ...	Material		<input checked="" type="checkbox"/>		
STEEL	PTC_THERMAL_EXPANSION_C...	Real Num...	0.000012		Full ...	Material		<input checked="" type="checkbox"/>	Thermal ...	/C
STEEL	PTC_SPECIFIC_HEAT	Real Num...	473340988.8...		Full ...	Material		<input checked="" type="checkbox"/>	Specific ...	mm^2/(se...
STEEL	PTC_THERMAL_CONDUCTIVITY	Real Num...	43012.523728		Full ...	Material		<input checked="" type="checkbox"/>	Thermal ...	mm kg /(s...
STEEL	PTC_MASS_DENSITY	Real Num...	0.000008		Full ...	Material		<input checked="" type="checkbox"/>	Density	kg/mm^3
STEEL	PTC_XHATCH_FILE	String	STEEL		Full ...	Material				
BEAM.PRT	E	Real Num...	199947961.4...		Locke...	Relation			Stress	Pa
BEAM.PRT	IYY	Real Num...	18455902.26...		Locke...	Relation			Area Mo...	m^4
BEAM.PRT	L	Real Num...	6000.000000		Locke...	Relation			Length	m
BEAM.PRT	X	Real Num...	4500.000000		Locke...	Relation			Length	m
SECOND_MOMENT	XSEC_AREA	Real Num...	2724.800000		Locke...	Analysis ...			Area	mm^2
SECOND_MOMENT	XSEC_IXX	Real Num...	1419344.810...		Locke...	Analysis ...			Area Mo...	mm^4
SECOND_MOMENT	XSEC_IYY	Real Num...	18455902.26...		Locke...	Analysis ...			Area Mo...	mm^4

Figure 18: Used parameters highlighted.

There is a *Unit* field in the parameter listing. So next we need to update Creo *Relations* to use the standard SI units (Table 2). Notice that FID_280 is changed to FID_SECOND_MOMENT (this how we can refer to all analysis).

Table 2: Updated Relations field. Notice the values.

IYY:MATHCAD = XSEC_IYY:FID_SECOND_MOMENT/10^12
L:MATHCAD = d15/1000
X:MATHCAD = d26/1000
E:MATHCAD = PTC_YOUNG_MODULUS:MTRL_40*1000

Go back to Mathcad and select **Update Inputs** (🔄) from *Integration* group. Then create a new **Text Block** (📄) called *Checks* and write

I	CTRL + -	yy	=	ENTER
---	----------	----	---	-------

This will show the value of I_{yy} variable. The value will be by default in m^4 , but we can change it by writing the missing **m** in the unit field (Figure 19).

Checks

$$I_{yy} = (1.846 \cdot 10^{-5}) \cdot mm^4$$

Figure 19: Added another m.

In the same way, show also E value (Figure 20). As you perhaps remember from the basic applied mechanics course(s), the modulus value of steel is about 200 GPa.

Inputs from Creo

$$L := 6 \text{ m}$$

$$E := 199947961488.167 \text{ Pa}$$

$$I_{yy} := 1.8459022666667E-05 \text{ m}^4$$

$$x := 4.5 \text{ m}$$

Checks

$$I_{yy} = (1.846 \cdot 10^{-7}) \text{ mm}^4$$

$$E = 199.948 \text{ GPa}$$

Other Inputs

$$F := 1 \text{ kN}$$

Figure 20: Current worksheet.

Save and Push the worksheet (CTRL+S).

Equation

We have all needed inputs, next we define equation that can be used to solve displacement. Create an equation as seen in Figure 21. Some useful hotkey commands can be seen in Table 3.

$$w_y(x) := \frac{F \cdot L^3}{6 \cdot E \cdot I_{yy}} \cdot \left(3 \cdot \frac{x^2}{L^2} - \frac{x^3}{L^3} \right)$$

Figure 21: Equation to calculate displacement in y-direction.

Table 3: Some useful hotkey commands.

Command	Meaning	Example
CTRL+-	subscript	w_{sub}
/	divider (use before to divide equation)	$\frac{\quad}{\quad}$
^	exponent	L^{exp}
*	multiplication	$F \cdot$
SPACE	moving wihtin equation (in the example SPACE was hitted several times to select all in the equation, so the next input will affect all higlighted sympols)	$\frac{F \cdot L^3}{6 \cdot E \cdot I_{yy}}$

Solving

To solve previously defined equation, write

w	CTRL+-	y	(x	=
---	--------	---	---	---	---

This will calculate the displacement in the location of x-parameter (X plane in Creo). Remember to change units to mm (default is m). To get the maximum displacement in the Y-direction, calculate displacement in L (Figure 22).

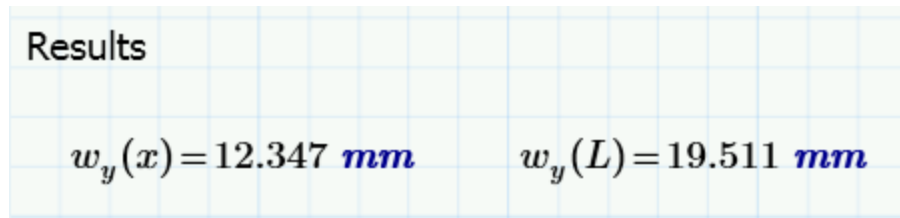


Figure 22: Displacement calculated in x and L locations.

Updating model

Go to Creo and change the offset value of X plane to **2500**. Then select **Update Inputs** (↺) from *Integration* group in **Input/Output** tab to update calculations. Save your Mathcad worksheet (**CTRL+S**).

Plotting

Next we create a plot to see how displacement is changed relative to the length. From **Plots** tab, select **XY Plot** (✚) from *Traces* group (Figure 23).

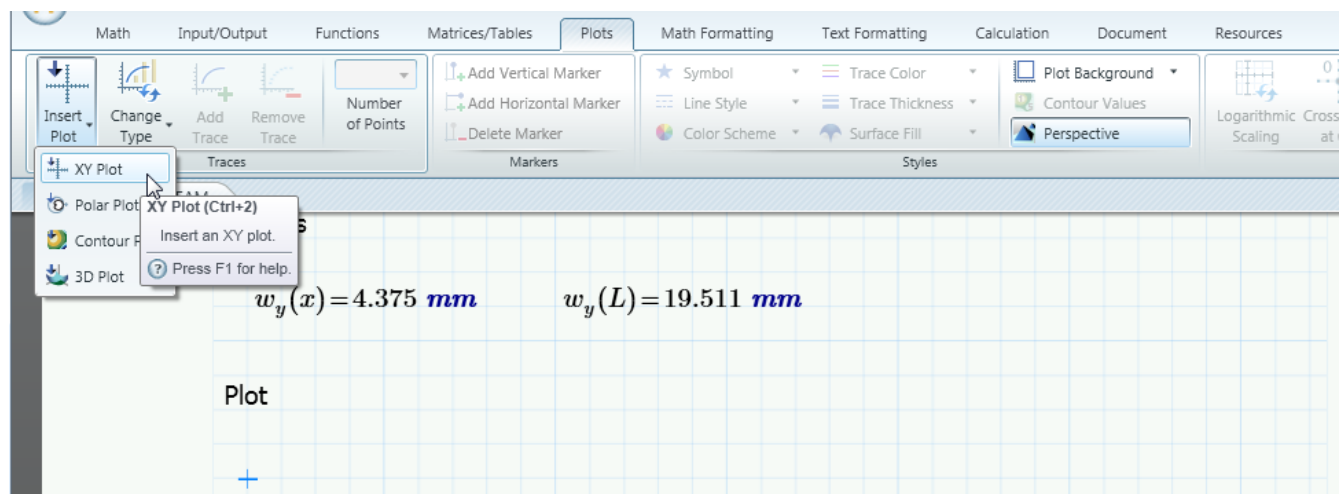


Figure 23: Inserting a XY plot.

An empty plot will appear to your worksheet. As *y-axis*, give $w_y(x)$ and **mm** as units. As *x-axis*, give **x** and **m** as units (Figure 24). You can resize the plot to make it wider.

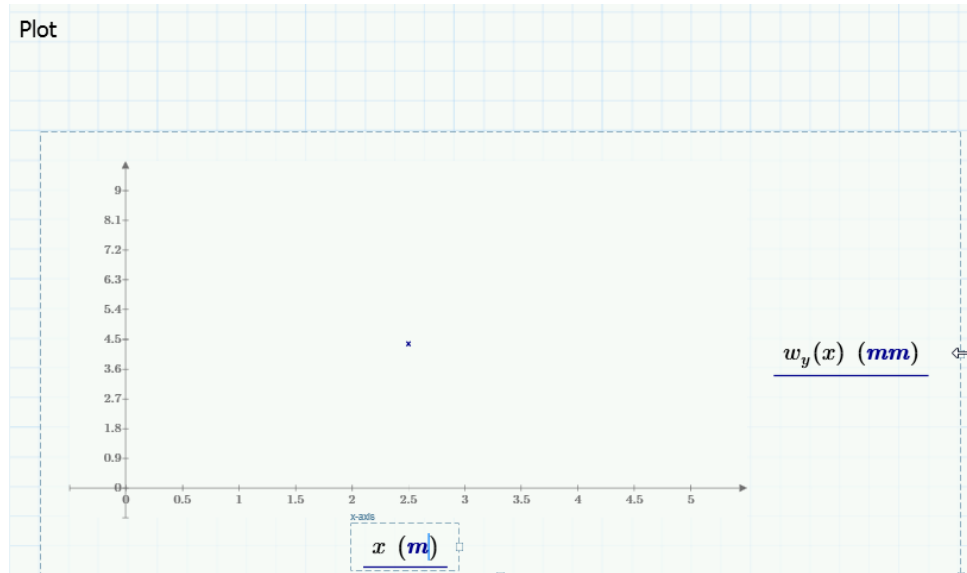


Figure 24: Created plot and resizing the plotting area.

Our plot plots only one point, because x is given a fixed value (2.5 m). To plot values from 0...6, we need to define x as a vector. Add before plotting area (\rightarrow is right arrow key)

x	:	0m	,	0.1m	\rightarrow	\rightarrow	L	ENTER
---	---	----	---	------	---------------	---------------	---	-------

Now the plot updates and shows how displacement is changed from 0 to L using 0.1 as a step (Figure 25). Step means, that values are calculated on w_y -function values 0, 0.1, 0.2, 0.3, ..., 6,0.

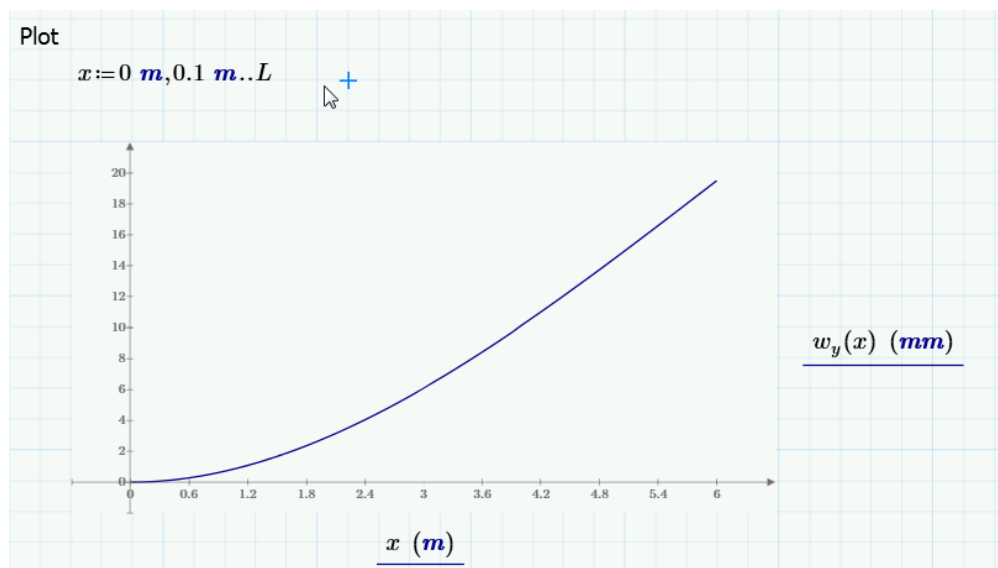


Figure 25: x variable updated.

Directions

Our displacement has positive values, because our coordinate axes and forces are defined as seen in Figure 26.



Figure 26: Our beam case before changing the direction of the force.

To change the direction of the force F , change it value to -1 kN . Now the graph updates and displacement makes more sense (Figure 27), because we want to calculate how much beam bends, if it needs to hold 1 kN load ($\sim 1000 \text{ kg}$).

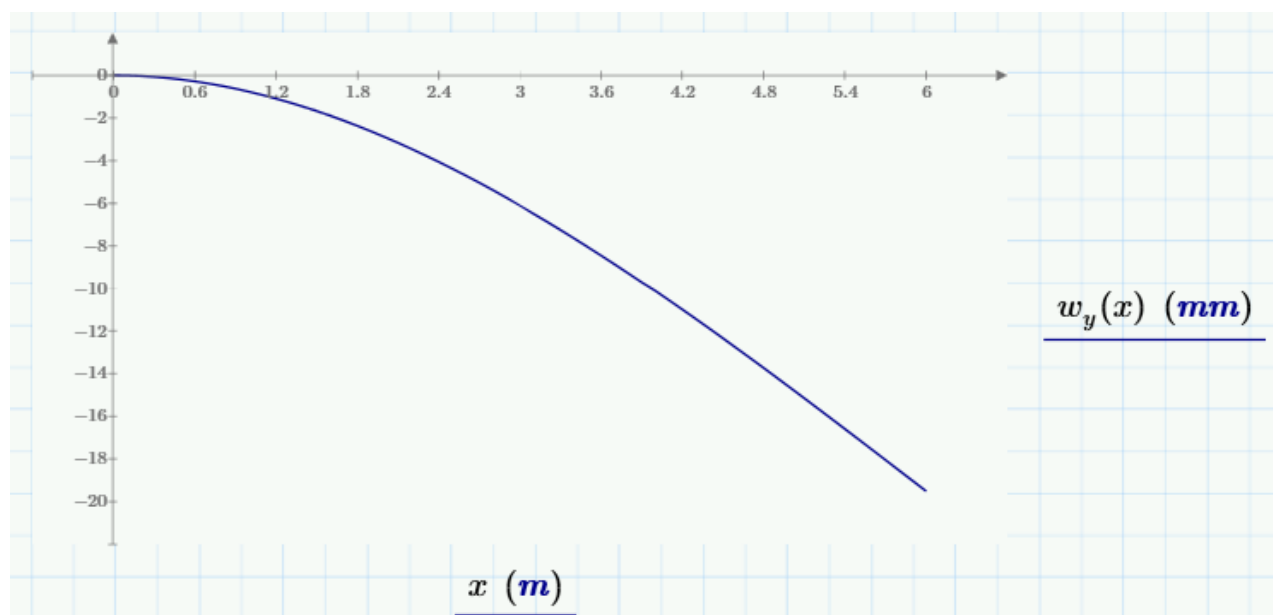


Figure 27: Updated displacement curve.

Define Outputs

To pass a value back to Creo, outputs can be used. Select $w_y(x)$ and $w_y(L)$, and from **Input/Output** tab, select **Assign Outputs** (🔌) from *Integration* group. A small *out* text appears to the top of selected field. Save your mode (**CTRL+S**) and go to Creo. In Creo, open **Parameters** (⌘) and look parameters of **Embedded Mathcad** (Figure 28).

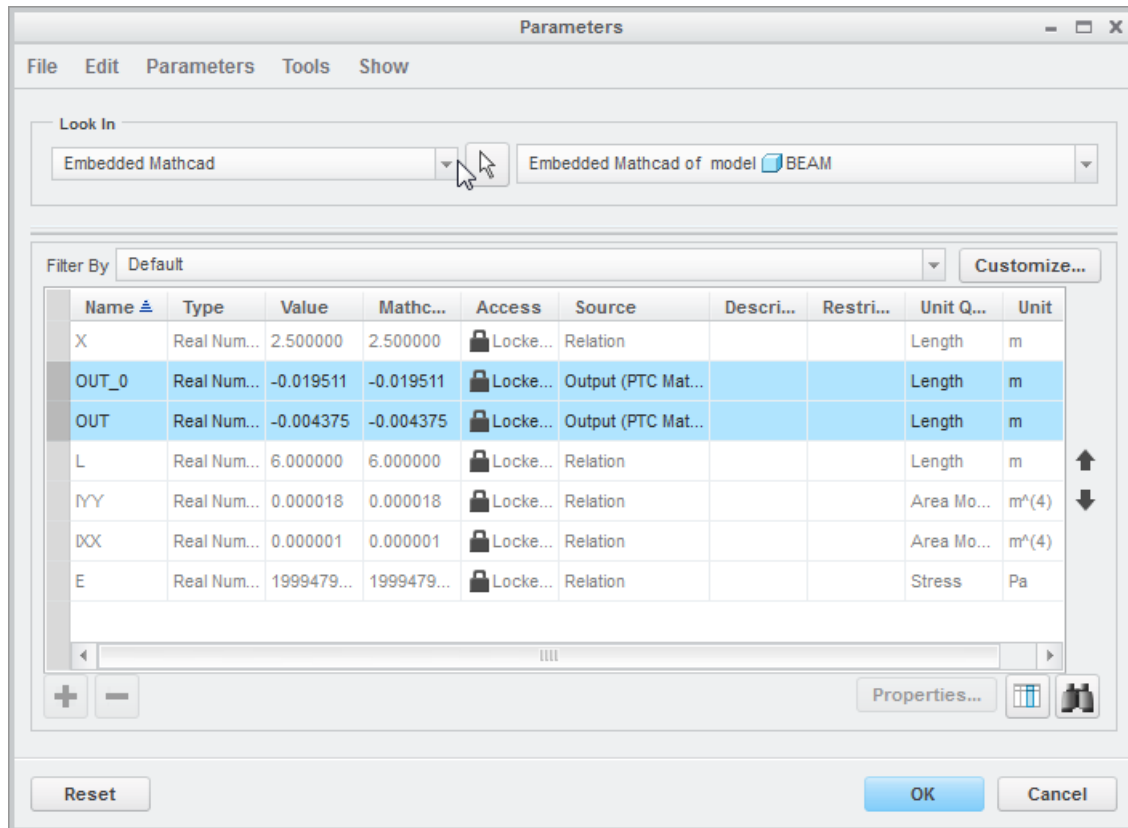


Figure 28: Parameters of the embedded Mathcad worksheet. Output parameters highlighted.

As you can see, parameter named OUT exists (displacement in the location of X plane). The naming policy is so, that the first output is OUT, second OUT_0, third OUT_1 and so on.

Creo Simulate – Strength simulations

From Mathcad calculations, we got **-19.5 mm** as maximum displacement in the y-direction. Next we perform a simple strength analysis to validate our calculations.

Select **Applications** → **Simulate** (🏗️, Figure 29).

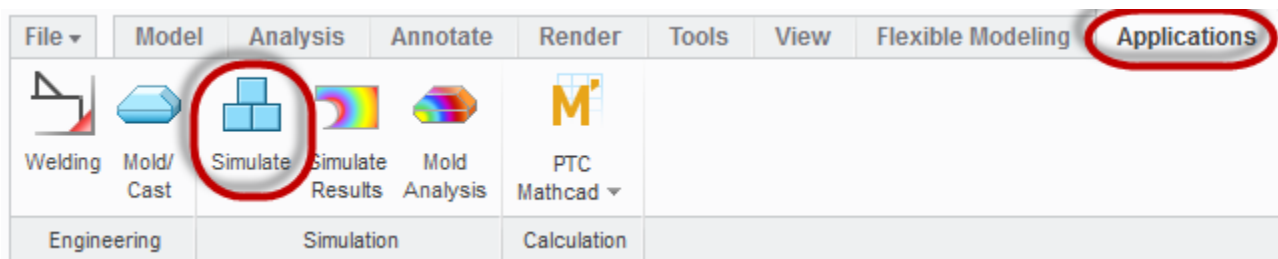



Figure 29: Simulate.

This opens Creo's integrated strength/thermal simulation application that uses FEM. It is common for CAD programs to have some integrated simulation mode for solving FEM models. There are also "pure" FEM programs, like Abaqus, that have more complex options to grate meshes and analyze solutions.

To perform a strength analysis, we need to define:

- loads
- constraints
- material.

Notice, that Creos ribbon-UI is built so, that you start from the left and going right you defined needed objects. Select **Force/Moment** () from *Loads* group and select the non-coordinate-system end (**Ctrl+D** to place part to default orientation if needed) of the beam as seen in Figure 30. Give value of **-1 kN** in the Y-direction (notice the small coordinate system in the bottom-right. Click **OK**.

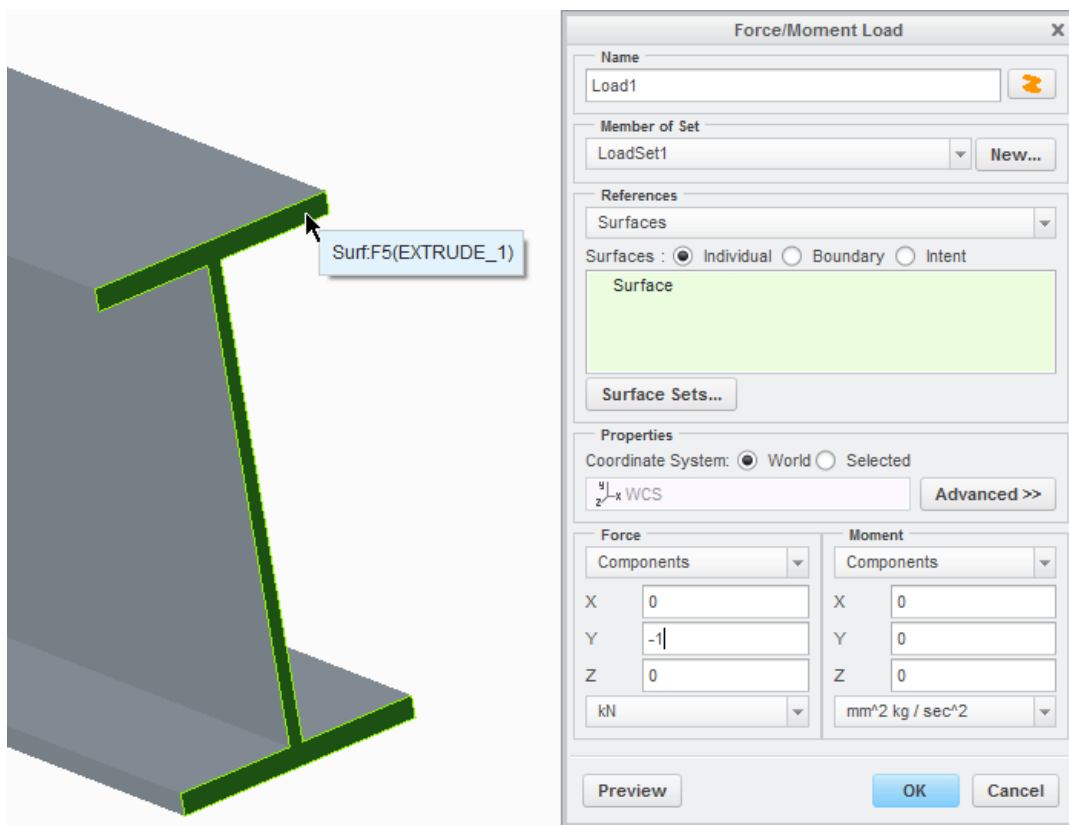


Figure 30: Selecting a surface to place -1 kN load.

Select **Displacement** (📏) from the *Constraints* group. Select the end other surface of the beam and accept all default values. This will fix this end to the ground (Figure 31). **OK** when ready.

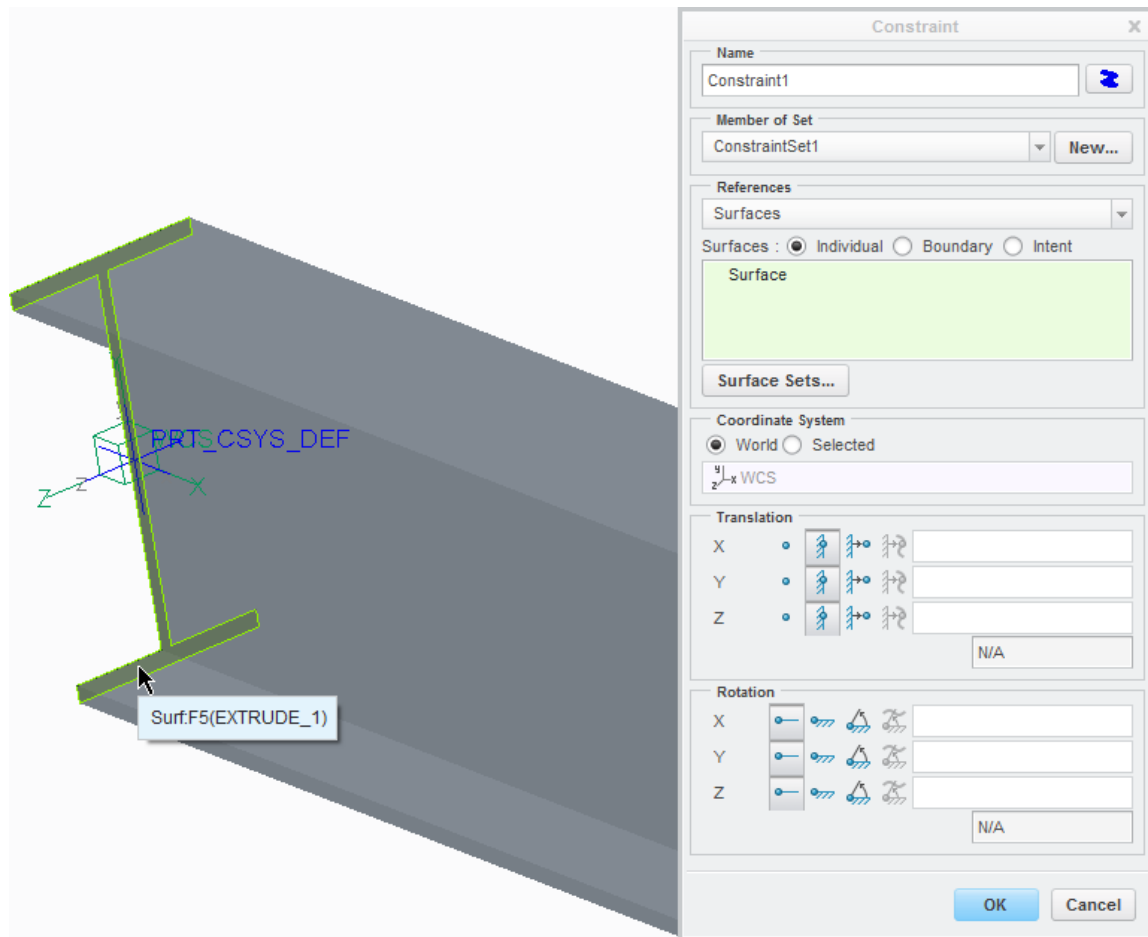


Figure 31: Surface selected for displacement constraint. *Csys display* is on.

Select **Material Assignment** (📁) from *Materials* group. Program will offer STEEL (because part template is using steel as a material), accept it (**OK**).

Now our simulation model is ready. Creo uses automatic meshing to create needed FEM mesh. We can of course redefine meshing options, but for a simple analysis the automatic mesh will be fine.

Select **Analysis and Studies** (📊) from *Run* group. A window appears. Select **File** → **New Static** (Figure 32) to create a new static analysis.

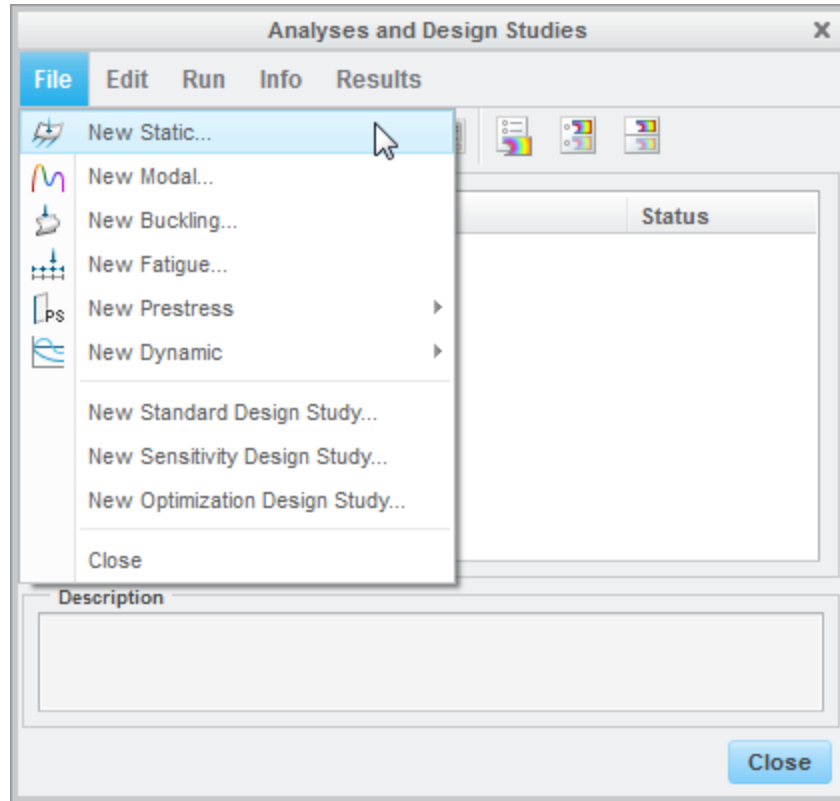


Figure 32: Creating a static analysis.

Default setting will be fine, just rename analysis1 as **Static_Y_Force_1kN** or similar. It is a good policy to rename simulations, because this will create a new folder to your working directory and it is easier later to return this analyses. Click **OK**. Press the green flag (🚩) to start. Answer **Yes** to the question and wait until simulation is done (takes about 1 min).

When simulation is ready, diagnostics window will appear. You can **Close** it. Select 📊 to review results. In **Quantity** tab, select from drop-down menus **Displacement** and **Y** as values. Check that others are similar as in Figure 33. Select **Display Options** tab and check options **Deformed** and **Show Element Edges**. Click **OK and Show** to see results.

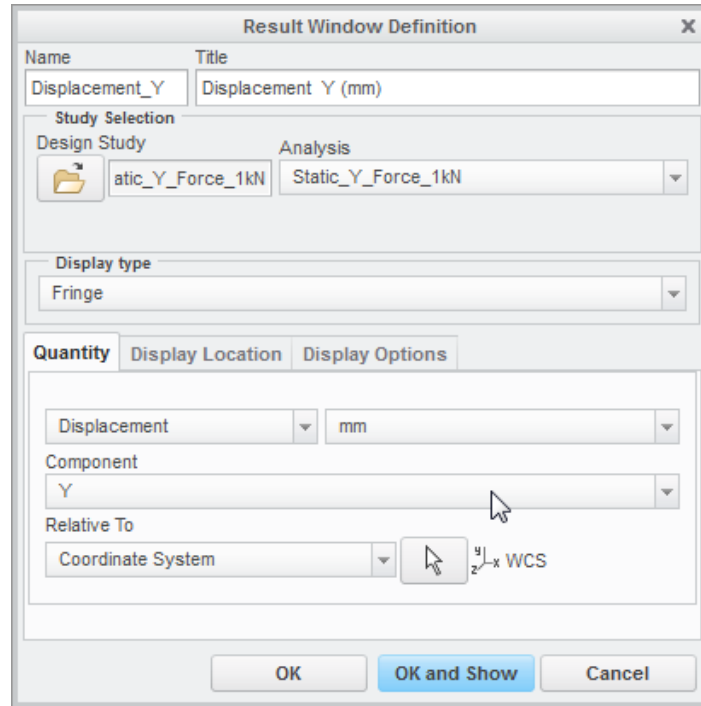


Figure 33: Values for a new result window. Notice the changed *Name* and *Title* fields.

You can read from the left that maximum displacement is about **19.6 mm** that is very close to the analytical solution (Figure 34).

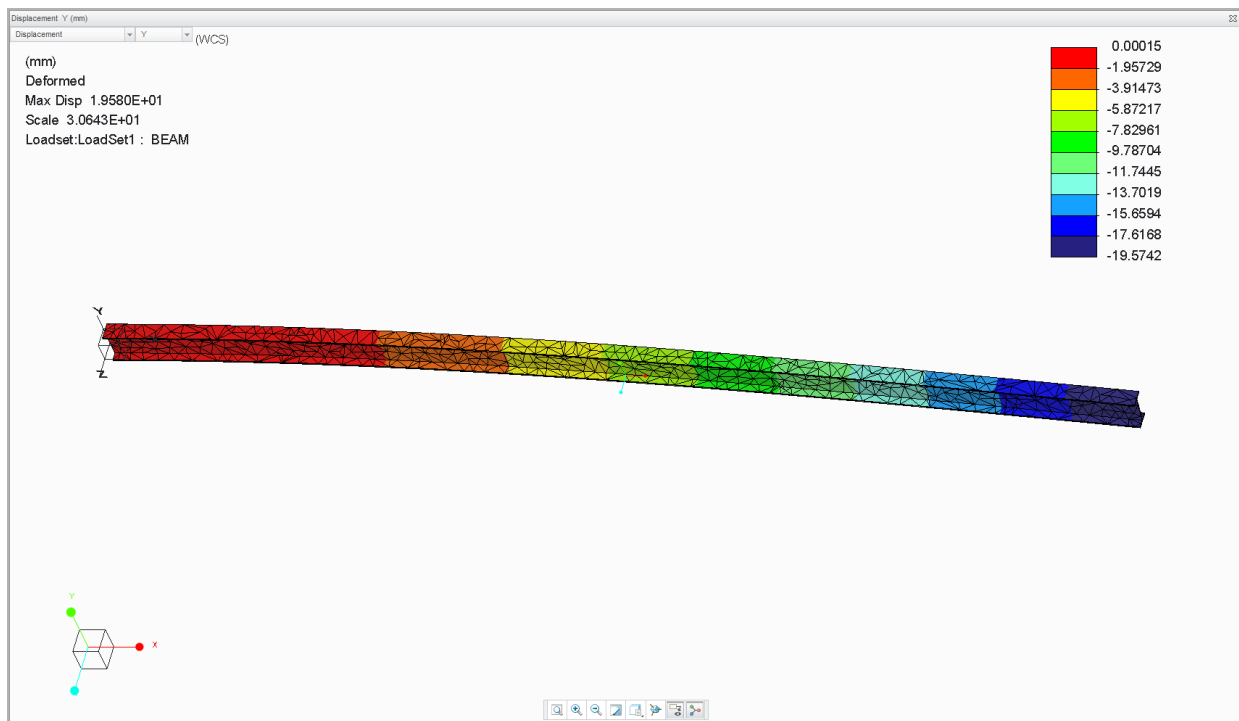




Figure 34: Result window.

Be free to check what other results you can get from FEM analysis by double-clicking the result window. You can for ex. check the von Mises stresses in the beam. When ready, close the result window and select option **Save**. You can then place result window (*rwd) to the simulation folder and give some good name (you can return to this view by selecting **Results**  → **Open**).

Close () the Simulate mode and save your model. This concludes this exercise.