ISSN 2395-1621

# Computer Aided Numerical Simulation of Over-Moulded Front End Carrier of Automobile

#1K. D. Lagwankar, #2P. D. Darade

[1]koustubh.lagwankar@gmail.com
[2]daradepd@gmail.com

#12Department of Mechanical Engineering, Sinhgad Institute of Technology and Science, S. P. Pune University,
Pune, India

**ABSTRACT**

**Reducing the weight of automobile components without compromising their strength is one of the greatest challenges in today's world, since the reduction in weight improves the performance of automobiles. This urge of weight reduction led to a thorough research in this field and development of new manufacturing methods. Metal reinforced plastics, produced using Over-moulding process are one of the effective technologies used widely in automobile industry. In this paper, only the central part of an over-moulded front end carrier is considered for study. Front end carrier is a structure which supports the radiator and hood-latch assembly. It is usually made up of sheet metal with C section and Crossed rib structure of plastic, where the introduction of the later improves its stiffness while keeping the weight minimum. Multiple methods for modelling the over-moulded part in ABAQUS software will be studied in this paper. Simulation results from these methods will be correlated with numerical calculations. The method with the best correlation will be suggested for Static simulation of the over-moulded part.**

ARTICLE INFO

**Article History**

Received:30th September 2015

Received in revised form :

2nd October 2015

Accepted:6th October, 2015

**Published online :**

**9th October 2015**

*Keywords*:- **Front End Carrier, Over-molded Plastic Parts, Static Simulation.**

## I. INTRODUCTION

Automobile industry is at constant need for finding out the different techniques to reduce the weight of the car without compromising its strength. Reduction in weight will improve performance of the vehicle as well as improves the fuel economy. Front End Carrier is a complex structure which holds the radiator assembly, Headlamp assembly and Hood latch assembly. The complete assembly is known as front end module. The creating module not only saves the assembly time but also reduce the investment cost. Front end module can be easily out sourced to other supplier thereby reducing the capital investment.

Polymer Metal Hybrid (PMH) technology is one of the solution for achieving the strength along with reduced weight. In this technology metal components are strengthen with plastic inserts. This method reduces weight without compromising the strength. It also helps designer to build the complex yet optimized structure. Plastics can easily be shaped into complex geometry unlike metal structures.

PMH technology uses advantages of both structures. Plastics used are generally Polypropylene (PP) or Polyamide (PA) reinforced with small glass-fibre having large compressive strength and Seel used generally of high grade which will have good tensile strength. In this way advantages of both materials are combined to achieve the great results.

The components are manufactured with the injection machine tool. Here the previously formed metal structure is kept in the core of injection tool and molten plastic is flown over it. This technology is also called as Over-moulding. This is very efficient process and no external fastening is required to join metal and plastic. Arrangements are made have a mechanical interlock between metal and plastic. It can be safely assumed that the interlock works perfect during its loading cycle and there is no slip between metal and plastic part.

Generally a C-Section sheet metal is inserted with plastic ribs. This combination provides excellent resistance to bending of beams.

In this paper we are considering the latch loads which will be acting on the front end carrier during its service life. The front end module is attached at its end to the BIW structure. Hence it is considered as simply supported beam with transverse load at its centre.

## II. LITERATURE REVIEW

Massimiliano BOCCIARELLI and Pierluigi COLOMBI have studied the behaviour of Elasto-plastic analysis of steel beams reinforced with Carbon Fibre Strip. In this paper a numerical procedure is used to evaluate the bending moment and shear force diagrams. The numerical treatment is done considering the principle of virtual work. A method based on cross-sectional equilibrium and strain compatibility is then proposed to predict the FRP axial stress and interface shear stress of the steel beam reinforced by FRP materials.[2]

M. Grujicic had published a research paper on injection over moulding of polymer-metal hybrid structures. This paper overviews PMH technology used to manufacture the automobile structure parts. Interaction between metal and plastic is studied in this paper. Also different technologies used to manufacture the PMH part are discussed. A computer simulation is done to optimise the loaded structure with plastic structure. This paper mainly overviews the load transfer between metal and polymer structure. [3]

S. C. NG and S. LEE have published their work under A Study of Flexural Behaviour of Reinforced Concrete Beam strengthened with Carbon Fibre-Reinforced Plastic (CFRP). This paper presents analytical study of flexural strength of reinforced beams. In this study, a simple and direct analytical procedure has been developed to evaluate the flexural capacity of concrete beams strengthened with CFRP and to predict their failure modes. A comparison between the analytical results and the data obtained from the literature has been made with good agreement. [4]

M. Grujicic et. al. have presented their work under title Suitability analysis of a polymer–metal hybrid technology based on high-strength steels and direct polymer-to-metal adhesion for use in load-bearing automotive body-in-white applications. In this paper, load transfer between stamped sheet-metal and injection-moulded riblike plastic subcomponents is accomplished through a variety of nanometre-to-micron scale chemical and mechanical phenomena which enable direct adhesion between the two materials. The resultant adhesion strength in a 5–10MPa range has been assessed. [5]

## III. NUMERICAL CALCULATION

For this work we will consider a small portion of over-moulded structural member of front end module. This module is manufactured with the stamped steel in C shape structure. Over-moulding is done to have a direct adhesion between injected rib structure and metal part. The crossed rib structure is chosen to have better flexural stiffness in transverse loading in both directions.

We will consider Euler-Bernoulli Beam theory for calculation of deflation of beam under loading condition.

$$\frac{d^2y}{dx^2} = \frac{M_x}{EI}$$

In this equation, y is the deflection beam in Y direction. $M_x$ corresponds to the bending moment along the length of beam and E is Young's Modulus and I is area moment of inertia of cross-section of beam. [1]

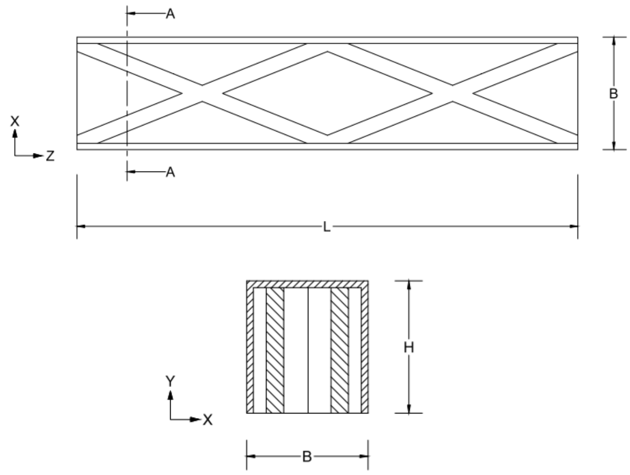


Fig.1 Over-Moulded beam with crossed rib structure

In above figure L represents the length of beam along Z direction, B is width of beam along X direction and H is height of beam along Y direction. Here the inner rib structure is made up of fibre reinforced plastic and outer body is made up of stamped plastic.

For this study we will transform the composite beam into an equivalent beam of one material. Here we will consider beam is made of metal structure alone. For this we will have to calculate the equivalent thickness of plastic beam when transformed to metal.

$$t_2 = \frac{E_{plastic}}{E_{metal}}\, t_{plastic}$$

Where, is equivalent thickness of plastic rib structure.

### A. *Loading of Beam in Y Direction*

We will consider first the loading of beam Y direction. We can show that varying distance between plastic ribs at different cross-sections does not have any effect on the location of neutral axis.

First we will calculate the area moment of inertia for any arbitrary section. We will divide this cross-section into 5 rectangular sections viz. $S_1$ to $S_5$ as shown in figure.

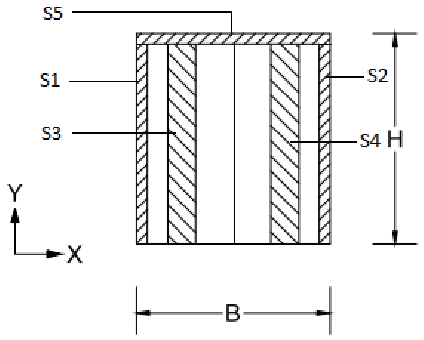


Fig.2 Sectional details for arbitrary cross section A-A

The location of neutral axis can be calculated with following formula.

$$\bar{y} = \frac{\sum y_i A_i}{\sum A_i} = \frac{2 \times \left(A_1 \times \frac{B - t_1}{2}\right) + 2 \times \left(A_2 \times \frac{B - t_2}{2}\right)}{A_1 + A_2 + A_3}$$

TABLE I
DETAILS FOR CALCUATION OF AREA MOMENT OF INERTIA IN XZ PLANE

| | Sec t-ion | Area | $y_i$ | B | H | $d_i$ |
|---|---|---|---|---|---|---|
| 1 | $S_1$ | $A_1$ | $H - t_1/2$ | $t_1$ | $H - t_1$ | $\bar{y} - y_1$ |
| 2 | $S_2$ | $A_1$ | $H - t_1/2$ | $t_1$ | $H - t_1$ | $\bar{y} - y_2$ |
| 3 | $S_3$ | $A_2$ | $H - t_2/2$ | $t_2$ | $H - t_2$ | $\bar{y} - y_3$ |
| 4 | $S_4$ | $A_2$ | $H - t_2/2$ | $t_2$ | $H - t_2$ | $\bar{y} - y_4$ |
| 5 | $S_5$ | $A_3$ | $H - t_1$ | B | $t_1$ | $\bar{y} - y_5$ |

$$I_i = \frac{BH^3}{12}$$

$$I = \sum I_i - A_i d_i$$

Deflection in such case can be calculated as:

$$Y = \frac{PL^3}{48EI}$$

Where, P is the load on the beam in negative Y direction, L is the length of beam along Z direction, E is young's modulus and I is area moment of inertia in XZ plane.

*B. Loading of beam in X direction*

In case of loading in X directions, the varying distance between plastic ribs at various cross-sections have effect on the calculation of area moment of inertia. Hence we have to derive the formula for calculation of deflection in X direction

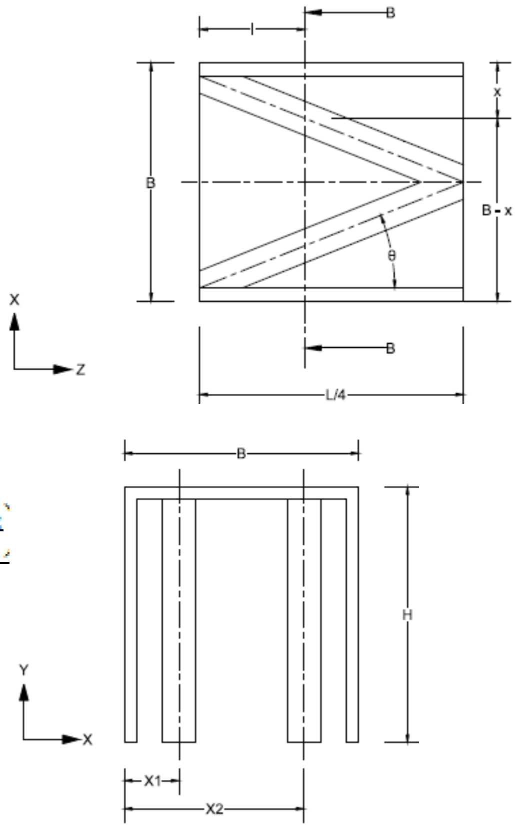


Fig.3 Portion of Beam for derivation

For this calculation we first have to find out the area moment of inertia. Here the value of I is constantly changing with cross-section. Hence we will have following derivation.

Consider a small section of beam having length L/4, breadth B and Height H. We will have a cross-section at a dist *l* from one end of beam. Let be the angle between rib and metal structure as shown in the figure. *x* and $x_1$ are the distances from the reference point for calculation I

$$x = l \tan\theta$$
$$x_1 = B - l \tan\theta$$

We can show that the value of neutral axis at every cross-section will be as follows,

$$\bar{x} = \frac{B}{2}$$

The area moment of inertia can be calculated by,

$$I_i = \frac{H}{1}$$

$$I = \sum I_i - A_i d_i$$

The different parameters used to calculate the area moment of inertia are listed in the table below.

TABLE 2
DETAILS FOR CALCULATION OF AREA MOMENT OF INERTIA IN YZ PLANE

| | Sect-ion | Area | $x_i$ | B | H | $d_i$ |
|---|---|---|---|---|---|---|
| 1 | $S_1$ | $A_1$ | 0 | $t_1$ | $H-t_1$ | $\lvert b/2\rvert$ |
| 2 | $S_2$ | $A_1$ | $l\tan\theta$ | $t_1$ | $H-t_1$ | $\lvert b/2-l\tan\theta\rvert$ |
| 3 | $S_3$ | $A_2$ | $B-l\tan\theta$ | $t_2$ | $H-t_2$ | $\lvert -b/2+l\tan\theta\rvert$ |
| 4 | $S_4$ | $A_2$ | B | $t_2$ | $H-t_2$ | $\lvert -b/2\rvert$ |
| 5 | $S_5$ | $A_3$ | $B/2$ | B | $t_1$ | $\lvert 0\rvert$ |

$$I_x = f(I_x) = 2I_1 + 2I_2 + 2I_3 + 2A_1\left(\frac{B}{2}\right)^2 + 2A_2\left|\frac{B}{2} - l\tan\theta\right|^2$$

Where $I_1$ to $I_3$ are area moment of inertia of different sections. $A_1$ to $A_3$ are the areas of different sections.

Bending moment equation for simply supported beam loaded centrally can be given as,

$$f(M_l) = \frac{1}{2}Pl - P(l - L/2)$$

According to Euler – Bernoulli Beam Deflection theory,

$$\frac{d^2x}{dl^2} = \frac{f(M_l)}{E.f(I_l)}$$

$$x = \frac{1}{E}\left[\int_{-L/2}^{L/2}\int_{-L/2}^{L/2}\frac{f(M_l)}{f(I_l)}\right]$$

From the equation above we can find out the deflection in X direction after loading.

As a case study we will consider the beam with length 500mm breadth 70 and height of 40mm. Thickness of sheet metal is 1mm where as that of plastic rib is 7 mm. The young's modulus for metal martial is 210000 N/mm$^2$ and young's modulus for plastic material is 1780 N/mm$^2$ The value of is considered as 9 $^0$.

With above specification the deflection of beam is calculated for load of 300N in Y direction and a load of 300 N in X direction. A small Matlab program is written to have the calculation and provide result

Following values of Deflection were obtained

$y = 0.040$ mm
$x = 0.075$ mm

## IV. CAE SIMULATIONS

A model is built in the HyperMesh software to such that it will co-relate with the numerical calculations. The model is built with 2 different materials. Red mesh is metal having C section where as mesh in blue is having plastic material with thickness as that in actual model. Two ends are supported with reaction in appropriate direction. Load of 300N is applied in X and Z direction respectively

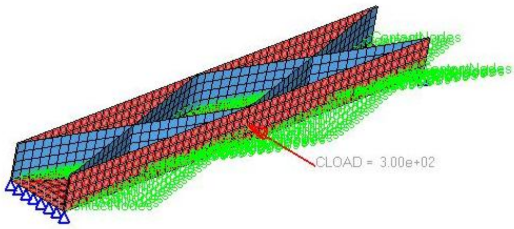


Fig.4 Simulation model for Calculation in ABAQUS

Different methods are used to join the plastic and sheet metal. One of them is by using KINCOUP connection (Rigid connection) between metal and plastic part. In this method KINCOUP element acts as rigid 1D element in Abaqus software. This method transfers load from one part to another. This modelling method is shown in figure below.

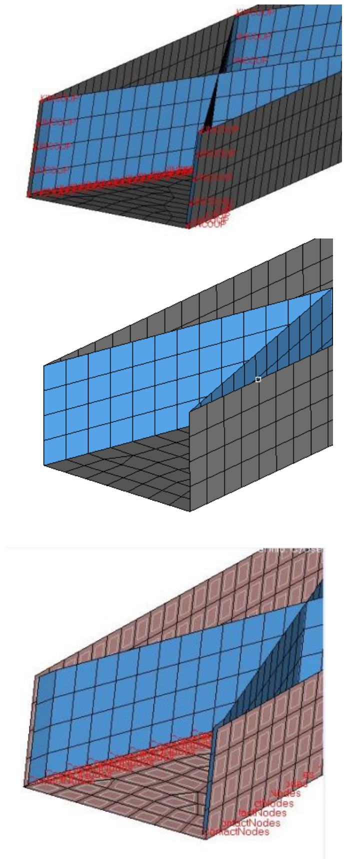

Fig.5 Different types of connections used to connect plastic and metal part in Abaqus solver

Other method is use of node to node connectivity between metal and plastic part. By use of this method a common calculation node is shared between metal part and plastic part. This approach acts as continuous system. This type of model is simple to build. Last method is use of ABAQUS contact of node to surface in the model between metal and plastic part. In this method a virtual surface is created by the solver. Due to deflection when nodes of one part comes in contact with each other then load is transferred to other surface [6]

This model is set up for the calculation in ABAQUS software and output is given for the nodal displacement.

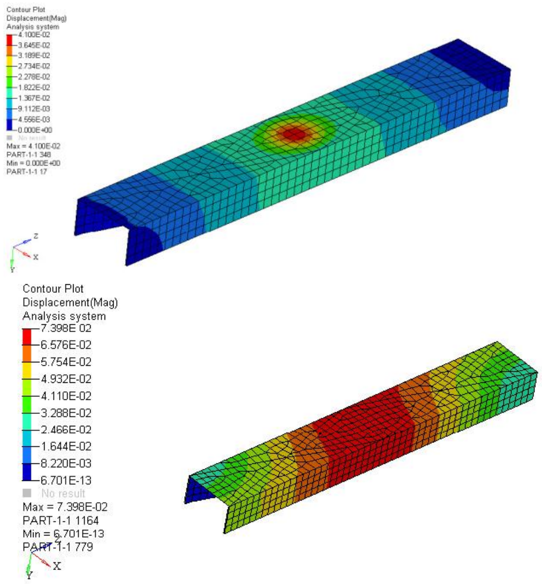


Fig.6 Simulation results after Calculation in ABAQUS with Y and X loading respectively

Results of all 6 simulations along with the calculations are enlisted in table below. We can see that the behaviour of the model with rigid KINCOUP connection is less stiffer than actual model. In this case the deflection of the model at given force is larger than the calculation. whereas behaviour of model with node to node connectivity show higher stiffness. In this case the deflection of model at given load is very low as compared with numerical calculation. Model working with contact has better correlation with the numerical results. In this case, the results of simulation show better correlation with numerical calculation.

TABLE 3
SIMULATION RESULTS WITH DIFFERENT METHODS

| **Method used** | **Deflection in Y in mm** | **Deflection in X in mm** |
|---|---|---|
| **Mathemeatical calculation** | **0.0400** | **0.0750** |
| **With rigid Connection** | **0.0567** | **0.0456** |
| **With Node to Node connectivity** | **0.0340** | **0.0623** |
| **With use of Contacts** | **0.0410** | **0.0739** |

## V. CONCLUSION

In this paper a calculation method is developed for composites beam made of polymer metal hybrid technology. Also the deflection of beam is calculated for Y and X loading respectively.

The result obtained in ABAQUS, where a tie contact between metal and plastic was applied, matches closely to the numerical results. Whereas other methods may under or over estimate the part stiffness. Hence we can conclude that over moulded plastics can be effectively analysed in ABAQUS by using tie contact methodology.

## VI.FUTURE SCOPE

A numerical method for calculation beam with plastic deformation of beam can be evaluated. This will allow for complete design of PMH beam.

Current method of using contacts for simulation can be improved with other parameters as well to correlate the results with physical test data.

## ACKNOWLEDGMENT

I am very thankful to my project guide Prof. P. D. Darade, Professor, Mechanical Engineering Department, Sinhgad Institute of Technology and Science, Pune, for his continuous support and encouragement in completing this work.

## REFERENCES

[1] S. Timoshenko and D. Young "Elements of Strength of Materials", 5th ed., New Delhi, Affiliated East-West Press Pvt. Ltd. Publishers, 2011.
[2] P. Bocciarelli, and P. Colombi "Elasto-Plastic Analysis of steel Beams Reinforced by Carbon Fibre Reinforced Polymer Strips," Engineering structures, vol. 29, No. 8, pp. 1808-1816, Nov. 2007.
[3] M. Grujicic, "Injection over molding of polymer-metal hybrid structures," American Journal of Science and Technology, vol. 1, No. 4, pp. 168-181, 2014.
[4] S. Ng and S. Lee "A Study of Flexural Behaviour of Reinforced Concrete Beam Strengthened with Carbon Fibre-Reinforced Plastic (CFRP)," Journal of Reinforced Plastics and Composites, vol. 21, pp. 95-105, 2002.
[5] M. Grujicic, V. Sellappan and S. Kotrika "Suitability Analysis of a polymer-metal hybrid technology based on high-strength steels and direct polymer-to-metal adhesion for use in load-bearing automotive body-in-white applications," Journal of Materials Processing Technology, vol. 5, pp. 40-56, 2008.
[6] H. Kuhlmann, P. Volgers and Z. Zang, "Application and CAE Simulation of Over Molded Short and Continuous Fibre Thermoplastic Composite," 13th International LS-Dyna Users Conference, Dearborn, Michigan, USA, June. 8, 2014.

 